

ANSYS Fluent Tutorial Guide



ANSYS, Inc. Southpointe 275 Technology Drive Canonsburg, PA 15317 ansysinfo@ansys.com http://www.ansys.com (T) 724-746-3304 (F) 724-514-9494 Release 15.0 November 2013



Copyright and Trademark Information

© 2013 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

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

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE 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. is certified to ISO 9001:2008.

U.S. Government Rights

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

Third-Party Software

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

Published in the U.S.A.

Table of Contents

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	xv
1. Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a	Mixing
Elbow	1
1.1. Introduction	
1.2. Prerequisites	2
1.3. Problem Description	
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	
1.4.6. Displaying Results in ANSYS Fluent and CFD-Post	
1.4.7. Duplicating the Fluent-Based Fluid Flow Analysis System	
1.4.8. Changing the Geometry in ANSYS DesignModeler	
1.4.9. Updating the Mesh in the ANSYS Meshing Application	
1.4.10. Calculating a New Solution in ANSYS Fluent	
1.4.11. Comparing the Results of Both Systems in CFD-Post	
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	
2.4.4. Defining Input Parameters in ANSYS Fluent and Running the Simulation	
2.4.5. Postprocessing and Setting the Output Parameters in ANSYS CFD-Post	
2.4.6. Creating Additional Design Points in ANSYS Workbench	
2.4.7. Postprocessing the New Design Points in CFD-Post	
2.4.8. 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	
3.4. Setup and Solution	
3.4.1. Preparation	
3.4.2. Launching ANSYS Fluent	
3.4.3. Reading the Mesh	
3.4.4. General Settings	
3.4.5. Models 3.4.6. Materials	
J.H.O. Materials	138

3.4.7. Cell Zone Conditions	141
3.4.8. Boundary Conditions	143
3.4.9. Solution	147
3.4.10. Displaying the Preliminary Solution	157
3.4.11. Using the Coupled Solver	172
3.4.12. Adapting the Mesh	
3.5. Summary	191
4. Modeling Periodic Flow and Heat Transfer	193
4.1. Introduction	193
4.2. Prerequisites	193
4.3. Problem Description	194
4.4. Setup and Solution	194
4.4.1. Preparation	195
4.4.2. Mesh	195
4.4.3. General Settings	198
4.4.4. Models	198
4.4.5. Materials	199
4.4.6. Cell Zone Conditions	201
4.4.7. Periodic Conditions	202
4.4.8. Boundary Conditions	
4.4.9. Solution	
4.4.10. Postprocessing	209
4.5. Summary	219
4.6. Further Improvements	
5. Modeling External Compressible Flow	
5.1. Introduction	221
5.2. Prerequisites	221
5.3. Problem Description	222
5.4. Setup and Solution	222
5.4.1. Preparation	
5.4.2. Mesh	223
5.4.3. General Settings	226
5.4.4. Models	226
5.4.5. Materials	227
5.4.6. Boundary Conditions	229
5.4.7. Operating Conditions	
5.4.8. Solution	232
5.4.9. Postprocessing	249
5.5. Summary	255
5.6. Further Improvements	255
6. Modeling Transient Compressible Flow	
6.1. Introduction	257
6.2. Prerequisites	257
6.3. Problem Description	258
6.4. Setup and Solution	
6.4.1. Preparation	258
6.4.2. Reading and Checking the Mesh	259
6.4.3. Specifying Solver and Analysis Type	261
6.4.4. Specifying the Models	263
6.4.5. Editing the Material Properties	264
6.4.6. Setting the Operating Conditions	265
6.4.7. Creating the Boundary Conditions	266

6.4.8. Setting the Solution Parameters for Steady Flow and Solving	
6.4.9. Enabling Time Dependence and Setting Transient Conditions	
6.4.10. Specifying Solution Parameters for Transient Flow and Solving	285
6.4.11. Saving and Postprocessing Time-Dependent Data Sets	288
6.5. Summary	
6.6. Further Improvements	
7. Modeling Radiation and Natural Convection	305
7.1. Introduction	305
7.2. Prerequisites	305
7.3. Problem Description	305
7.4. Setup and Solution	306
7.4.1. Preparation	306
7.4.2. Reading and Checking the Mesh	307
7.4.3. Specifying Solver and Analysis Type	308
7.4.4. Specifying the Models	
7.4.5. Defining the Materials	
7.4.6. Specifying Boundary Conditions	314
7.4.7. Obtaining the Solution	318
7.4.8. Postprocessing	324
7.4.9. Comparing the Contour Plots after Varying Radiating Surfaces	339
7.4.10. S2S Definition, Solution, and Postprocessing with Partial Enclosure	351
7.5. Summary	355
7.6. Further Improvements	356
8. Using the Discrete Ordinates Radiation Model	357
8.1. Introduction	357
8.2. Prerequisites	357
8.3. Problem Description	358
8.4. Setup and Solution	358
8.4.1. Preparation	359
8.4.2. Mesh	359
8.4.3. General Settings	360
8.4.4. Models	363
8.4.5. Materials	364
8.4.6. Cell Zone Conditions	366
8.4.7. Boundary Conditions	368
8.4.8. Solution	376
8.4.9. Postprocessing	380
8.4.10. Iterate for Higher Pixels	388
8.4.11. Iterate for Higher Divisions	392
8.4.12. Make the Reflector Completely Diffuse	400
8.4.13. Change the Boundary Type of Baffle	402
8.5. Summary	404
8.6. Further Improvements	404
9. Using a Non-Conformal Mesh	405
9.1. Introduction	405
9.2. Prerequisites	405
9.3. Problem Description	406
9.4. Setup and Solution	407
9.4.1. Preparation	407
9.4.2. Mesh	408
9.4.3. General Settings	411
9.4.4. Models	412

9.4.5. Materials	. 414
9.4.6. Cell Zone Conditions	. 414
9.4.7. Operating Conditions	. 416
9.4.8. Boundary Conditions	. 416
9.4.9. Mesh Interfaces	. 426
9.4.10. Solution	428
9.4.11. Postprocessing	. 431
9.5. Summary	. 443
9.6. Further Improvements	. 443
10. Modeling Flow Through Porous Media	. 445
10.1. Introduction	445
10.2. Prerequisites	. 445
10.3. Problem Description	. 446
10.4. Setup and Solution	. 446
10.4.1. Preparation	
10.4.2. Mesh	. 447
10.4.3. General Settings	. 449
10.4.4. Models	. 450
10.4.5. Materials	
10.4.6. Cell Zone Conditions	
10.4.7. Boundary Conditions	
10.4.8. Solution	
10.4.9. Postprocessing	
10.5. Summary	
10.6. Further Improvements	
11. Using a Single Rotating Reference Frame	
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. Solution Using the Standard k- $arepsilon$ Model	
11.4.9. Postprocessing for the Standard k- ε Solution	
11.4.10. Solution Using the RNG k- ε Model	
11.4.11. Postprocessing for the RNG k- ε Solution	
11.5. Summary	
11.6. Further Improvements	
11.7. References	
12. Using Multiple Reference Frames	
12.1. Introduction	
12.2. Prerequisites	
12.3. Problem Description	
12.4. Setup and Solution	
12.4.1. Preparation	
12.4.2. Reading and Checking the Mesh and Setting the Units	
12.4.3. Specifying Solver and Analysis Type	

12.4.4. Specifying the Models	
12.4.5. Specifying Materials	527
12.4.6. Specifying Cell Zone Conditions	. 529
12.4.7. Setting Boundary Conditions	. 532
12.4.8. Defining Mesh Interfaces	. 534
12.4.9. Obtaining the Solution	. 535
12.4.10. Step 9: Postprocessing	. 541
12.5. Summary	. 547
12.6. Further Improvements	. 548
13. Using the Mixing Plane Model	. 549
13.1. Introduction	549
13.2. Prerequisites	. 549
13.3. Problem Description	. 549
13.4. Setup and Solution	. 550
13.4.1. Preparation	. 550
13.4.2. Mesh	. 551
13.4.3. General Settings	. 551
13.4.4. Models	. 554
13.4.5. Mixing Plane	. 555
13.4.6. Materials	
13.4.7. Cell Zone Conditions	. 558
13.4.8. Boundary Conditions	
13.4.9. Solution	
13.4.10. Postprocessing	. 578
13.5. Summary	
13.6. Further Improvements	
14. Using Sliding Meshes	
14.1. Introduction	. 587
14.2. Prerequisites	. 587
14.3. Problem Description	
14.4. Setup and Solution	
14.4.1. Preparation	
14.4.2. Mesh	
14.4.3. General Settings	. 589
14.4.4. Models	
14.4.5. Materials	
14.4.6. Cell Zone Conditions	
14.4.7. Boundary Conditions	
14.4.8. Operating Conditions	
14.4.9. Mesh Interfaces	
14.4.10. Solution	
14.4.11. Postprocessing	
14.5. Summary	
14.6. Further Improvements	. 630
15. Using Dynamic Meshes	
15.1. Introduction	
15.2. Prerequisites	
15.3. Problem Description	
15.4. Setup and Solution	
15.4.1. Preparation	
15.4.2. Mesh	
15.4.3. General Settings	

15.4.4. Models	. 636
15.4.5. Materials	. 637
15.4.6. Boundary Conditions	. 639
15.4.7. Solution: Steady Flow	. 644
15.4.8. Time-Dependent Solution Setup	
15.4.9. Mesh Motion	
15.4.10. Time-Dependent Solution	655
15.4.11. Postprocessing	. 665
15.5. Summary	. 669
15.6. Further Improvements	
16. Modeling Species Transport and Gaseous Combustion	. 671
16.1. Introduction	. 671
16.2. Prerequisites	
16.3. Problem Description	
16.4. Background	. 672
16.5. Setup and Solution	. 672
16.5.1. Preparation	. 673
16.5.2. Mesh	. 673
16.5.3. General Settings	
16.5.4. Models	
16.5.5. Materials	
16.5.6. Boundary Conditions	. 683
16.5.7. Initial Reaction Solution	
16.5.8. Postprocessing	
16.5.9. NOx Prediction	
16.6. Summary	
16.7. Further Improvements	
17. Using the Non-Premixed Combustion 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 Checking the Mesh	
17.4.3. Specifying Solver and Analysis Type	
17.4.4. Specifying the Models	
17.4.5. Defining Materials and Properties	
17.4.6. Specifying Boundary Conditions	
17.4.7. Specifying Operating Conditions	
17.4.8. Obtaining Solution	
17.4.9. Postprocessing	
17.4.10. Energy Balances Reporting	
17.5. Summary	
17.6. References	
17.7. Further Improvements	
18. Modeling Surface Chemistry	
18.1. Introduction	
18.2. Prerequisites	
18.3. Problem Description	
18.4. Setup and Solution	
18.4.1. Preparation	
18.4.2. Reading and Checking the Mesh	

18.4.3. Specifying Solver and Analysis Type	
18.4.4. Specifying the Models	765
18.4.5. Defining Materials and Properties	767
18.4.6. Specifying Boundary Conditions	776
18.4.7. Setting the Operating Conditions	782
18.4.8. Simulating Non-Reacting Flow	783
18.4.9. Simulating Reacting Flow	786
18.4.10. Postprocessing the Solution Results	793
18.5. Summary	801
18.6. Further Improvements	801
19. Modeling Evaporating Liquid Spray	803
19.1. Introduction	
19.2. Prerequisites	803
19.3. Problem Description	803
19.4. Setup and Solution	804
19.4.1. Preparation	
19.4.2. Reading the Mesh	
19.4.3. General Settings	
19.4.4. Specifying the Models	
19.4.5. Materials	
19.4.6. Boundary Conditions	
19.4.7. Initial Solution Without Droplets	
19.4.8. Create a Spray Injection	
19.4.9. Solution	
19.4.10. Postprocessing	
19.5. Summary	
19.6. Further Improvements	
20. Using the VOF Model	
20.1. Introduction	
20.2. Prerequisites	
20.3. Problem Description	
20.4. Setup and Solution	
20.4.1. Preparation	
20.4.2. Reading and Manipulating the Mesh	
20.4.3. General Settings	
20.4.4. Models	
20.4.5. Materials	
20.4.6. Phases	
20.4.7. Operating Conditions	
20.4.8. User-Defined Function (UDF)	
20.4.9. Boundary Conditions	
20.4.10. Solution	
20.4.11. Postprocessing	
20.5. Summary	
20.5. Summary	
21. Modeling Cavitation	
21.1. Introduction	
21.2. Prerequisites	
21.2. Problem Description	
21.3. Problem Description	
21.4. Setup and Solution	
21.4.2. Reading and Checking the Mesh	
21.7.2. NEAULIY AND CHECKINY UNE MEDI	073

	21.4.3. General Settings	895
	21.4.4. Models	896
	21.4.5. Materials	898
	21.4.6. Phases	901
	21.4.7. Boundary Conditions	904
	21.4.8. Operating Conditions	909
	21.4.9. Solution	
	21.4.10. Postprocessing	914
	21.5. Summary	
	21.6. Further Improvements	
22.	Using the Mixture and Eulerian Multiphase Models	
	22.1. Introduction	
	22.2. Prerequisites	
	22.3. Problem Description	
	22.4. Setup and Solution	
	22.4.1. Preparation	
	22.4.2. Mesh	
	22.4.3. General Settings	
	22.4.4. Models	
	22.4.5. Materials	
	22.4.6. Phases	
	22.4.7. Boundary Conditions	
	22.4.8. Operating Conditions	
	22.4.9. Solution Using the Mixture Model	
	22.4.10. Postprocessing for the Mixture Solution	
	22.4.11. Higher Order Solution using the Mixture Model	
	22.1.1 1.1 lighter ofder Solution using the Mixture Model	
	22.4.12 Setup and Solution for the Fulerian Model	945
	22.4.12. Setup and Solution for the Eulerian Model	
	22.4.13. Postprocessing for the Eulerian Model	950
	22.4.13. Postprocessing for the Eulerian Model	950 953
23	22.4.13. Postprocessing for the Eulerian Model	950 953 954
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 954 955
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 954 955 955
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 954 955 955 955
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 954 955 955 955 955
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 954 955 955 955 956 956
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 955 955 955 955 956 956
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 954 955 955 955 956 956 956
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 955 955 955 956 956 956 957 958
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 955 955 955 956 956 956 957 958 965
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 955 955 955 956 956 956 957 958 965 967
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 955 955 955 956 956 956 957 958 965 967 969
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 955 955 955 956 956 956 957 958 965 967 969 972
23.	22.4.13. Postprocessing for the Eulerian Model	950 953 955 955 955 956 956 956 957 958 967 969 967 969 972 973
23.	22.4.13. Postprocessing for the Eulerian Model 22.5. Summary 22.6. Further Improvements Using the Eulerian Multiphase Model for Granular Flow 23.1. Introduction 23.2. Prerequisites 23.3. Problem Description 23.4. Setup and Solution 23.4.1. Preparation 23.4.2. Mesh 23.4.3. General Settings 23.4.4. Models 23.4.5. Materials 23.4.6. Phases 23.4.7. User-Defined Function (UDF) 23.4.8. Cell Zone Conditions 23.4.9. Solution	950 953 955 955 955 956 956 956 956 957 958 965 965 967 969 972 973 976
23.	22.4.13. Postprocessing for the Eulerian Model 22.5. Summary 22.6. Further Improvements Using the Eulerian Multiphase Model for Granular Flow 23.1. Introduction 23.2. Prerequisites 23.3. Problem Description 23.4. Setup and Solution 23.4.1. Preparation 23.4.2. Mesh 23.4.3. General Settings 23.4.4. Models 23.4.5. Materials 23.4.5. Materials 23.4.6. Phases 23.4.7. User-Defined Function (UDF) 23.4.8. Cell Zone Conditions 23.4.10. Postprocessing	950 953 955 955 955 956 956 956 956 957 958 967 969 972 973 976 990
23.	22.4.13. Postprocessing for the Eulerian Model22.5. Summary22.6. Further ImprovementsUsing the Eulerian Multiphase Model for Granular Flow23.1. Introduction23.2. Prerequisites23.3. Problem Description23.4. Setup and Solution23.4.1. Preparation23.4.2. Mesh23.4.3. General Settings23.4.4. Models23.4.5. Materials23.4.6. Phases23.4.7. User-Defined Function (UDF)23.4.8. Cell Zone Conditions23.4.10. Postprocessing23.5. Summary	950 953 955 955 955 956 956 956 957 958 965 967 969 972 973 976 990 994
	22.4.13. Postprocessing for the Eulerian Model22.5. Summary22.6. Further ImprovementsUsing the Eulerian Multiphase Model for Granular Flow23.1. Introduction23.2. Prerequisites23.3. Problem Description23.4. Setup and Solution23.4.1. Preparation23.4.2. Mesh23.4.3. General Settings23.4.4. Models23.4.5. Materials23.4.6. Phases23.4.7. User-Defined Function (UDF)23.4.8. Cell Zone Conditions23.4.10. Postprocessing23.5. Summary23.6. Further Improvements	950 953 955 955 955 956 956 956 956 957 958 967 967 969 972 973 976 990 994 994
	22.4.13. Postprocessing for the Eulerian Model 22.5. Summary 22.6. Further Improvements Using the Eulerian Multiphase Model for Granular Flow 23.1. Introduction 23.2. Prerequisites 23.3. Problem Description 23.4. Setup and Solution 23.4.1. Preparation 23.4.2. Mesh 23.4.3. General Settings 23.4.4. Models 23.4.7. User-Defined Function (UDF) 23.4.8. Cell Zone Conditions 23.4.9. Solution 23.4.10. Postprocessing 23.4.10. Postprocessing	950 953 955 955 955 956 956 956 956 957 958 965 967 969 972 973 976 990 994 994
	22.4.13. Postprocessing for the Eulerian Model 22.5. Summary 22.6. Further Improvements Using the Eulerian Multiphase Model for Granular Flow 23.1. Introduction 23.2. Prerequisites 23.3. Problem Description 23.4. Setup and Solution 23.4.1. Preparation 23.4.2. Mesh 23.4.3. General Settings 23.4.4. Models 23.4.5. Materials 23.4.6. Phases 23.4.7. User-Defined Function (UDF) 23.4.8. Cell Zone Conditions 23.4.9. Solution 23.4.10. Postprocessing 23.5. Summary 23.6. Further Improvements Modeling Solidification 24.1. Introduction	950 953 955 955 955 956 956 956 956 957 958 967 969 972 973 976 990 994 995 995
	22.4.13. Postprocessing for the Eulerian Model 22.5. Summary 22.6. Further Improvements Using the Eulerian Multiphase Model for Granular Flow 23.1. Introduction 23.2. Prerequisites 23.3. Problem Description 23.4. Setup and Solution 23.4.1. Preparation 23.4.2. Mesh 23.4.3. General Settings 23.4.4. Models 23.4.5. Materials 23.4.7. User-Defined Function (UDF) 23.4.8. Cell Zone Conditions 23.4.9. Solution 23.4.10. Postprocessing 23.4.5. Summary 23.4.5. Materials 23.4.6. Phases 23.4.7. User-Defined Function (UDF) 23.4.8. Cell Zone Conditions 23.4.9. Solution 23.4.10. Postprocessing 23.5. Summary 23.6. Further Improvements Modeling Solidification 24.1. Introduction 24.2. Prerequisites	950 953 955 955 955 956 956 956 957 958 967 958 967 969 972 973 976 990 994 995 995 995
	22.4.13. Postprocessing for the Eulerian Model 22.5. Summary 22.6. Further Improvements Using the Eulerian Multiphase Model for Granular Flow 23.1. Introduction 23.2. Prerequisites 23.3. Problem Description 23.4. Setup and Solution 23.4.1. Preparation 23.4.2. Mesh 23.4.3. General Settings 23.4.4. Models 23.4.5. Materials 23.4.6. Phases 23.4.7. User-Defined Function (UDF) 23.4.8. Cell Zone Conditions 23.4.9. Solution 23.4.10. Postprocessing 23.5. Summary 23.6. Further Improvements Modeling Solidification 24.1. Introduction	950 953 955 955 955 956 956 956 957 958 965 967 969 972 973 976 994 995 995 995 995

24.4.1. Preparation	
24.4.2. Reading and Checking the Mesh	998
24.4.3. Specifying Solver and Analysis Type	999
24.4.4. Specifying the Models	1001
24.4.5. Defining Materials	1002
24.4.6. Setting the Cell Zone Conditions	1004
24.4.7. Setting the Boundary Conditions	1005
24.4.8. Solution: Steady Conduction	1014
24.4.9. Solution: Transient Flow and Heat Transfer	1024
24.5. Summary	1035
24.6. Further Improvements	1035
25. Using the Eulerian Granular Multiphase Model with Heat Transfer	1037
25.1. Introduction	1037
25.2. Prerequisites	1037
25.3. Problem Description	
25.4. Setup and Solution	
25.4.1. Preparation	
25.4.2. Mesh	
25.4.3. General Settings	
25.4.4. Models	
25.4.5. UDF	
25.4.6. Materials	
25.4.7. Phases	
25.4.8. Boundary Conditions	
25.4.9. Solution	
25.4.10. Postprocessing	
25.5. Summary	
25.6. Further Improvements	
25.7. References	
26. Postprocessing	
26.1. Introduction	
26.2. Prerequisites	
26.3. Problem Description	
26.4. Setup and Solution	
26.4.1. Preparation	
26.4.2. Reading the Mesh	
26.4.3. Manipulating the Mesh in the Viewer	
26.4.4. Adding Lights	
26.4.5. Creating Isosurfaces	
26.4.6. Generating Contours	
26.4.7. Generating Velocity Vectors	
26.4.8. Creating Animation	
26.4.9. Displaying Pathlines	
26.4.10. Overlaying Velocity Vectors on the Pathline Display	
26.4.11. Creating Exploded Views	
26.4.12. Animating the Display of Results in Successive Streamwise Planes	
26.4.13. Generating XY Plots	
26.4.14. Creating Annotation	
26.4.15. Saving Hardcopy Files	
26.4.16. Generating Volume Integral Reports	
26.5. Summary	
27. Parallel Processing	

27.1. Introduction	29
27.2. Prerequisites	29
27.3. Problem Description	30
27.4. Setup and Solution	30
27.4.1. Preparation 113	30
27.4.2. Starting the Parallel Version of ANSYS Fluent 113	
27.4.2.1. Multiprocessor Machine 113	31
27.4.2.2. Network of Computers 113	32
27.4.3. Reading and Partitioning the Mesh 113	35
27.4.4. Solution 114	
27.4.5. Checking Parallel Performance 114	42
27.4.6. Postprocessing 114	43
27.5. Summary 114	46

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. 123)

and that you are familiar with the ANSYS Fluent navigation pane and menu 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. 123)

All of the tutorials include some postprocessing instructions, but Postprocessing (p. 1075) 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.

- 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.
- ANSYS 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.

Using This Manual

- Fluent Theory Guide contains reference information for how the physical models are implemented in Fluent.
- Fluent UDF 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.
- ANSYS Fluent Meshing Tutorials contains a number of examples of general mesh-generation techniques used in ANSYS Fluent Meshing.

Tutorials for release 15.0 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.
- ANSYS Fluent Meshing Text Command List contains a brief description of each of the commands in Fluent's meshing mode text interface.
- Fluent Adjoint Solver Module Manual 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.
- Fluent Battery Module Manual contains information about the background and usage of Fluent's Battery Module that allows you to analyze the behavior of electric batteries.
- Fluent Continuous Fiber Module Manual 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.
- Fluent Fuel Cell Modules Manual 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.
- Fluent Magnetohydrodynamics (MHD) Module Manual 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.
- Fluent Population Balance Module Manual 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 Fluent Under LSF contains information about using Fluent with Platform Computing's LSF software, a distributed computing resource management tool.
- Running Fluent Under PBS Professional contains information about using Fluent with Altair PBS Professional, an open workload management tool for local and distributed environments.

• Running Fluent Under SGE contains information about using Fluent with Sun Grid Engine (SGE) 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 Tutorials Postprocessing (p. 1075) and Parallel Processing (p. 1129) use existing case and data files.)

Some of the more complex tutorials may require a significant amount of computational time. If you want to look at the results immediately, without waiting for the calculation to finish, final solution files are provided in a solution_files folder that you can access after extracting the tutorial input archive.

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. 723).

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. 1075), 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. 723).

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. 1075), 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 the text of the tutorials to facilitate your learning process.

• Different type styles are used to indicate graphical user interface menu items and text interface menu items (e.g., **Zone Surface** dialog box, surface/zone-surface command).

- The text interface type style is also used when illustrating exactly what appears on the screen or exactly what you must type in the text window or in a dialog box.
- Instructions for performing each step in a tutorial will appear in standard type. Additional information about a step in a tutorial appears in italicized type.
- A mini flow chart is used to guide you through the navigation pane, which leads you to a specific task page or dialog box. For example,

4 Models $\rightarrow \equiv$ Multiphase \rightarrow Edit...

indicates that **Models** is selected in the navigation pane, which then opens the corresponding task page. In the **Models** task page, **Multiphase** is selected from the list. Clicking the **Edit...** button opens the **Multiphase** dialog box.

Also, a mini flow chart is used to indicate the menu selections that lead you to a specific command or dialog box. For example,

Define \rightarrow **Injections...**

indicates that the Injections... menu item can be selected from the Define pull-down menu.

The words surrounded by boxes invoke menus (or submenus) and the arrows point from a specific menu toward the item you should select from that menu.

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.

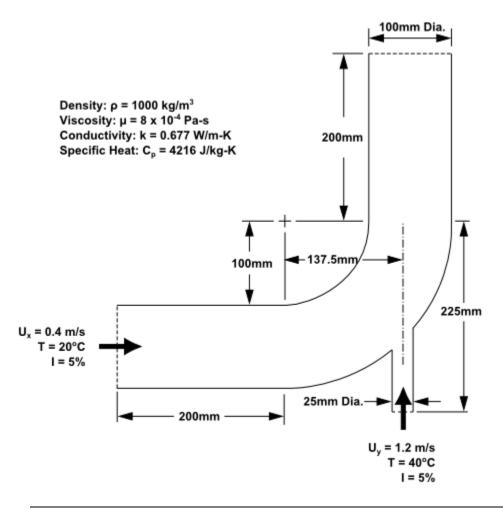
1.3. Problem Description

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

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. Displaying Results in ANSYS Fluent and CFD-Post
- 1.4.7. Duplicating the Fluent-Based Fluid Flow Analysis System
- 1.4.8. Changing the Geometry in ANSYS DesignModeler
- 1.4.9. Updating the Mesh in the ANSYS Meshing Application
- 1.4.10. Calculating a New Solution in ANSYS Fluent
- 1.4.11. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip elbow-workbench_R150.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

Tip

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. Start ANSYS Workbench by clicking the Windows **Start** menu, then selecting the **Workbench 15.0** option in the **ANSYS 15.0** program group.

Start \rightarrow All Programs \rightarrow ANSYS 15.0 \rightarrow Workbench 15.0

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.

Note

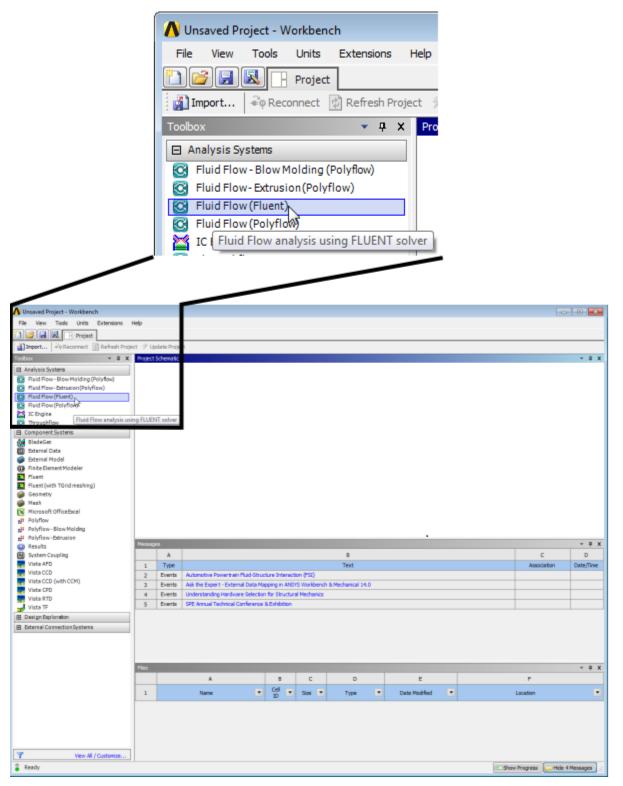
When you first start ANSYS Workbench, the **Getting Started** pop-up window is displayed, offering assistance through the online help for using the application. You can keep the window open, or close it by clicking the 'X' icon in the upper right-hand corner. If you need to access the online help at any time, use the **Help** menu, or press the **F1** key.

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						
File View Tools Units Extensions H	lelp					
🞦 🚅 🛃 📑 Project						
Import 🖓 Reconnect 🕼 Refresh Project 🍼 Update Project						
Toolbox 👻 🕂 X Project Schematic						
Analysis Systems						
🔀 Fluid Flow - Blow Molding (Polyflow)						
G Fluid Flow-Extrusion(Polyflow)						
🔇 Fluid Flow (Fluent)	1 💽 Fluid Flow (Fluent)					
🚱 Fluid Flow (Polyflow)	2 🥪 Geometry 📪					
🞽 IC Engine	3 🍘 Mesh 💡					
C Throughflow	4 🎡 Setup 👕					
Component Systems						
🚰 BladeGen	5 🕼 Solution 🔗 🖌					
External Data	6 🌍 Results 🛛 😨 🛓					
i External Model	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 high-lighted).
 - b. Enter elbow for the name of the analysis system.
- 4. Save the project.
 - a. Select the **Save** option under the **File** menu in ANSYS Workbench.

File \rightarrow 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 (²) 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.
- **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 ([≁]) 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 Up-to-Date.
- Interrupted (*) 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**.
- 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** (⁷/₂) 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 Understanding Cell States.

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 Files View.

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, 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.

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

In the ANSYS Workbench **Project Schematic**, double-click the **Geometry** cell in the elbow fluid flow analysis system. This displays the ANSYS DesignModeler application.

Tip

You can also right-click the **Geometry** cell to display the context menu, then select **New Geometry...**

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.

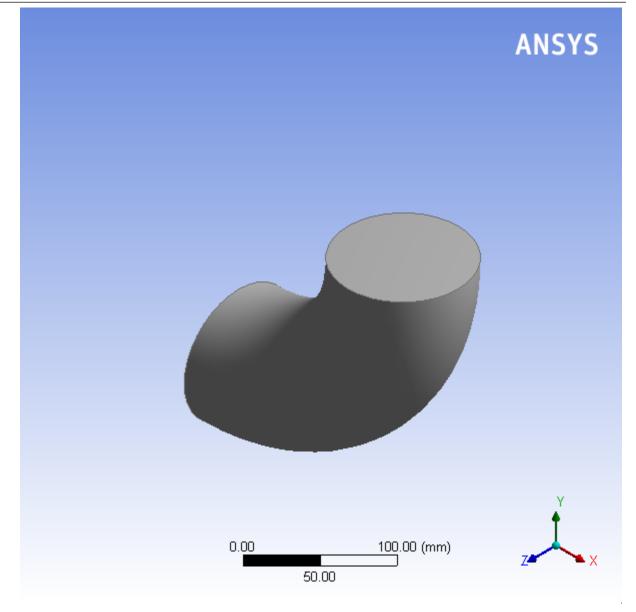
ii. In the **Details View** for the new torus (**Torus1**), set **Base Y Component** to -1 by clicking the **1** to the right of **FD10**, **Base Y Component**, entering -1, and pressing **Enter**. This specifies the direction vector from the origin to the center of the circular cross-section at the 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.

tails View	
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 ^{3/2} Generate</sup> that is located in the ANSYS DesignModeler toolbar.



- 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 **INTERCENT INTERCENT INTERCENT** 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**.

😳 A: elbow - DesignN	lodeler			
File Create Conce	pt Tools Units View	Help		
	DUndo @Redo	Select 🐄 📴 🖻 🐚 🚭 🗮 🥪 🗶 🗶 🛇 🔆	0 0 0 0 0 0 0 × 11 × 6 •	19
	1. A. A. K. J			
		🖇 📔 🍠 Generate 🖤 Share Topology 🔛 Parameters 📗 🌉 Extr	ude - Reuches - Suman - Stind off	
			aute Mikeopive Saveep SkiryLort	
		Sice 🗍 🚸 Point 📳 Conversion		
Tree Outline	ą.	Graphics		. .
E 🦓 A: elbow				
XYPlan				ANSYS
				111313
Torus1				
1 Part, 1				
	,			
Sketching Modeling				
Details View				
E Details of Extrude1				
Extrude	Extrude1			
Geometry	1 Face			
Operation	Add Material			
Direction Vector	Face Normal			
Direction	Normal			
Extent Type	Fixed			
FD1, Depth (>0)				
As Thin/Surface?	No	7 T		~
Merge Topology?	Yes	• •		
Face	1	• •		T
			100.00 (
		0.00	100.00 (mm)	Z X
			50.00	
		Model View Print Preview		
Continue Continue	··· Select direction vecto		1 Face: Area = 7854 mm ² Milli	meter Degree 0 0
Principle Creation	1 ··· Select direction vecto		1 Face: Area = 70.34 mm Milli	meter begree 0 0 //

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 .		
	ept Tools View Help		
		፦ 1011610000- 0+ Ⅲ 5+00000000- × *	© • 12
■+ ■+ //+ /	h• h• h• h• # #		
XVPlane 💌 🕯	🖡 None 👻 ಶ Gene	rate 👹 Share Topology 😥 Parameters	
Extrude 💏 Rev	olve 🐁 Sweep 🚯 Skin/Loft 🛛 🛅 1	Thin/Surface 💊 Blend 🔻 💊 Chamfer 🌆 Slice 🛛 🚸 Point 🚦 Conversion	
Tree Outline		Graphics	7
A: elbow			
XVPlan	ie .		ANCVC
ZXPlan			ANSYS
	ie		
🦽 Torus1			
💽 Extrude			
🚊- 📌 🔂 1 Part,	1 Body		
Sketching Modeling		4	
	0		
Details View	0		
Details of Extrude1			
Extrude	Extrude1		
Geometry	1 Face		
Operation	Add Material		
Direction Vector	Face Normal		
Direction	Normal		
Extent Type	Fixed		
FD1, Depth (>0)			
As Thin/Surface?	No		
Merge Topology?	Ves		
 Geometry Selection Face 	1		
rate	4		
			Y
			+
		0.00 100.00 (mm)	
			2 X
		50.00	
		Model View Print Preview	
Ready		No Selection	Millimeter 0 0 //
		1	

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:

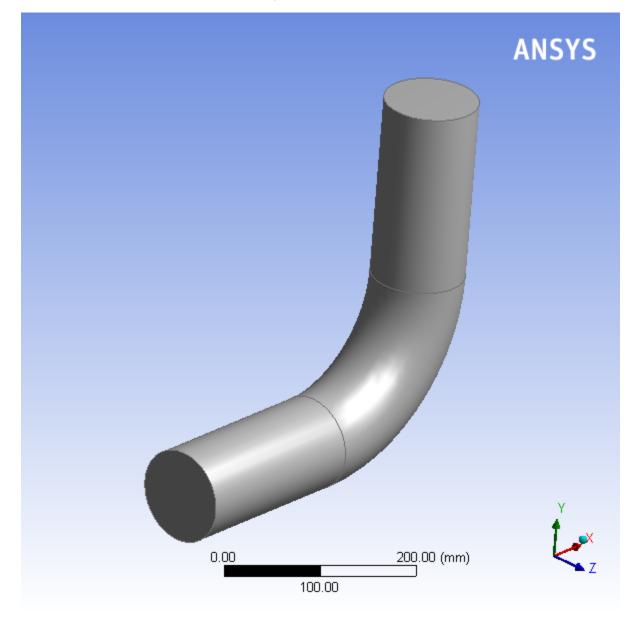
Action	Using Graphics Toolbar Buttons and the Mouse
Rotate view (vertical, hori- zontal)	After clicking the Rotate icon, , press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.
Translate or pan view	After clicking the Pan 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 Zoom icon, ^(Q) , press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.

Action	Using Graphics Toolbar Buttons and the Mouse
Box zoom	After clicking the Box Zoom icon, \bigoplus , 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, (a), 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. 15).

Figure 1.5: Elbow Main Pipe Geometry

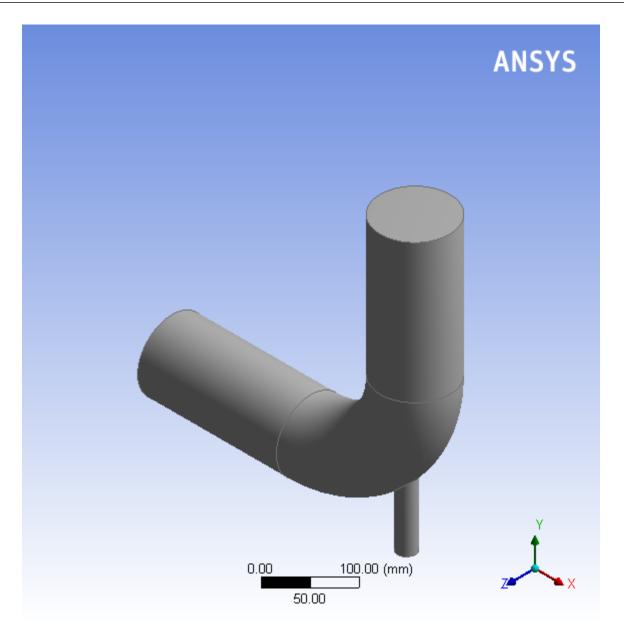


b. Next you will use a cylinder primitive to create the side pipe.

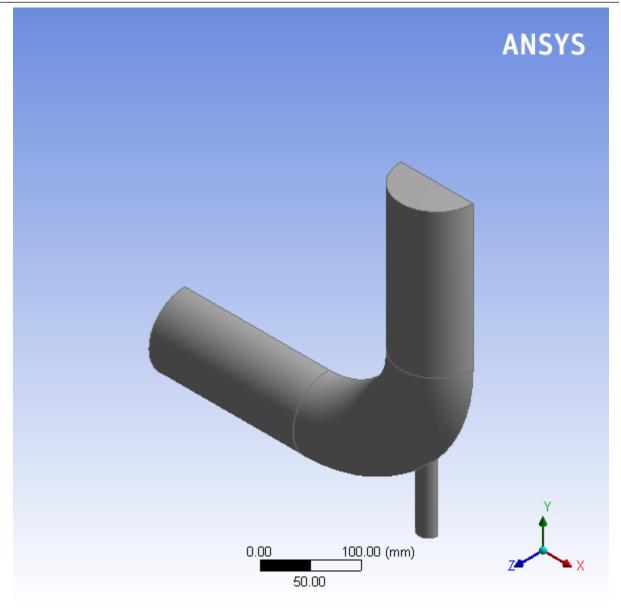
- i. Choose **Create** \rightarrow **Primitives** \rightarrow **Cylinder** from the menubar.
- ii. In the Details View, set the parameters for the cylinder as follows and click Generate:

Tab	Setting	Value
Details of Cylin-	BasePlane	XYPlane
der1	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
	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.

De	etails View	д
Ξ	Details of Body	
	Body	Fluid
	Volume	2.5159e+006 mm3
	Surface Area	1.7636e+005 mm ²
	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 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 Workb	ench Files View for the Pr	oject After Creating the Geometry
-------------------------	----------------------------	-----------------------------------

Files						
	А	В	С	D	E	
1	Name 💌	Cell 🔽 ID	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-woi
4	🂐 designPoint.wbdp		26 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wor

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	S Fluid Flow (Fluent)	
2	🔞 Geometry	× .
3	🎯 Mesh	2 🖌
4	🍓 Setup	? 🖌
5	Solution	? 🖌
6	🥩 Results	? 🖌
	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.

	M CFD]
File Edit View Units Tools	Help 🛛 🕂 💈 Generate Mesh 🏥 👪 🛆 🎯 🕶 🌒 Worksheet in
	<u>□</u> • • • • • • • • • • • • • • • • • • •
	Edge Coloring • $h \cdot h $
Model Dirtual Topology	Symmetry 🛛 🎕 Connections 🛛 🎕 Fracture 🛛 🎕 Mesh Numbering 🛛 🗐 Named Selection
Outline 4	
Filter: Name 💌	ANSYS
Project Model (A3) Model (A3) Coordinate Systems Mesh	
Details of "Model"	
Lighting	
Ambient 0.1	
Lighting Ambient 0.1 Diffuse 0.6	
Ambient 0.1	
Ambient 0.1 Diffuse 0.6 Specular 1	0.000 0.200 (m)
Ambient 0.1 Diffuse 0.6 Specular 1	Y T

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

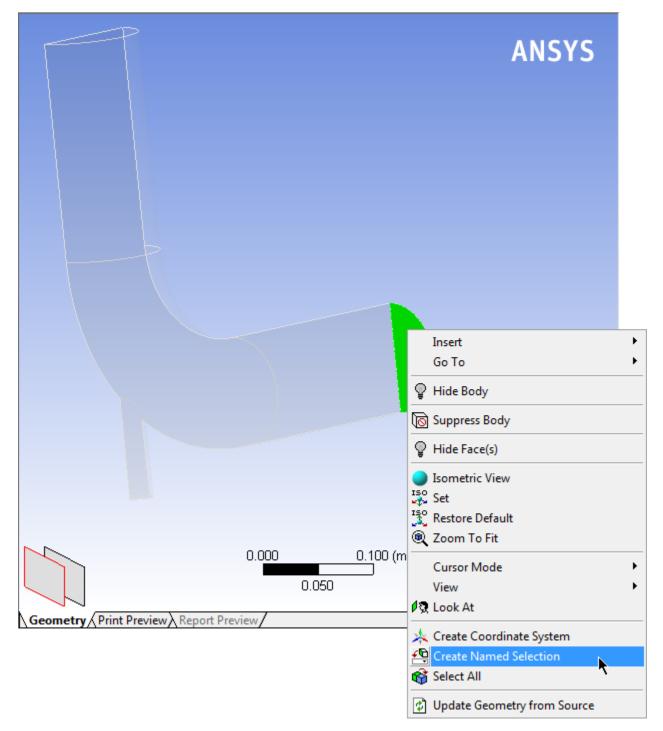
Use the Graphics Toolbar buttons and the mouse to manipulate the image until you can easily see the pipe openings and surfaces.

Tip

To select the inlet, the Single select ($\[b]$) 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
🗆 Туре
Location X
Location Y
Location Z
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).

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** (
 - 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**.

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

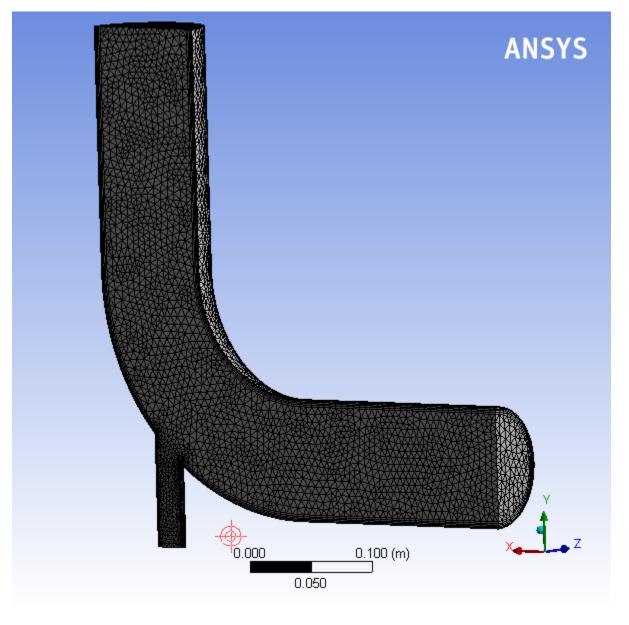
		ANSY
Insert	•	🚳 Method
Go To	•	🔍 Sizing
📁 Generate Mesh On Selected Bodies		Contact Sizing
 Preview Surface Mesh On Selected Bodies Clear Generated Data On Selected Bodies		A Inflation
Parts	•	-
P Hide Body		-
Suppress Body		-
Isometric View		
ISO Set		
Iso Restore Default		
🔍 Zoom To Fit		_
Cursor Mode	•	
View	•	v
🖉 Look At		L L L
oreate Coordinate System		•
Create Named Selection		Xand
😭 Select All		

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

- iv. Click the new Body Sizing control in the Outline tree.
- v. Enter 6e-3 for Element Size and press Enter.
- d. Click again on **Mesh** in the **Outline** view and expand the **Inflation** node in the **Details of "Mesh"** view 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 Creat	ion
--	-----

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, in this step you will set up a CFD analysis using ANSYS Fluent, then review the list of files generated by ANSYS Workbench.

1. Start ANSYS Fluent.

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

Iluent Launcher (Setting Edit Only)	
ANSYS	Fluent Launcher
Dimension 2D 3D	Options Double Precision Meshing Mode
Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme Do not show this panel again	Processing Options Serial Parallel
	ancel <u>H</u> elp 🔻

a. 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.

- i. Ensure that Serial from the Processing Options list is enabled.
- ii. Select Double Precision under Options.
- iii. Enable the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options.

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.

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

Note

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

·		
	villia64.win.ansys.com [3d, dp, pbns, lam] [ANSYS CFD	
File Mesh Define So	lve Adapt Surface Display Report Parallel V	ew Help
1	⑧ [[] 中 @ ⊕ ∥ @ 火 + +	■ + @ + <mark>設 # A 亞</mark> 聖
Meshing	General	1: Mash 🔻
Mesh Generation	Mesh	ANSYS
Solution Setup Senera Models Materials	Scale Check Report Quality Display	
Phases	Solver	
Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Type Velocity Formulation @ Pressure-Based @ Absolute Density-Based © Relative	
Solution	Steady	
Solution Methods Solution Controls Monitors	(1) Transient	
Solution Initialization Calculation Activities Run Calculation	Gravity Units	7
Results	Halp	Wesh
Graphics and Animations Plots		ANSYS Fluent (3d, dp, pons, lam)
Reports		Preparing mesh for display ^
		<pre>Writing Settings file "C:\Vork\Tut1\elbaw-workbench_files\dp@\FFF witing rp variables Done. writing fluid (type fluid) (mixture) Done. writing fluid (type fluid) (mixture) Done. writing uall-fluid (type interior) (mixture) Done writing uall-fluid (type wall) (mixture) Done. writing velocity-inlet-large (type velocity-inlet) (nixtu writing pressure-outlet (type pressure-outlet) (nixture) writing synmetry (type synmetry) (mixture) Done. writing zones nap name-id Done.</pre>

Figure 1.13: The ANSYS Fluent Application

2. Set general settings for the CFD analysis.

Note

Select **General** in the navigation pane to perform the mesh-related activities and to choose a solver.

⇔General

Meshing	General
Mesh Generation	Mesh
Solution Setup General Models Materials Phases	Scale Check Report Quality Display
Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Type Velocity Formulation Image: Pressure-Based Image: Pressure-Based Image: Density-Based Image: Pressure-Based Image: Time Time
Solution	Steady
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Transient Gravity Units
Results	Help
Graphics and Animations Plots Reports	

a. 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.

General → Units...

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

Set Units			×
Quantities kinematic-viscosity length-inverse length-time-inverse mag-permeability mass mass-diffusivity mass-flow mass-flow mass-flow mass-flux mass-transfer-rate mole-transfer-rate		Units m cm mm in ft Factor 0.001 Offset 0	Set All to default si british cgs
	New	Close Help	

- i. Select length in the Quantities list.
- ii. Select **mm** in the **Units** list.
- iii. 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.

b. Check the mesh.

\bigcirc General \rightarrow 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.

c. Review the mesh quality.

General → Report Quality

Note

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

Mesh Quality:

```
Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.
Minimum Orthogonal Quality = 2.54267e-01
Maximum Aspect Ratio = 2.18098e+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 Mesh Quality in the User's Guide.

3. Set up your models for the CFD simulation.

Models

Models
Models
Multiphase - Off
Energy - Off
Viscous - Laminar
Radiation - Off
Heat Exchanger - Off Species - Off
Discrete Phase - Off
Solidification & Melting - Off
Acoustics - Off
Eulerian Wall Film - Off
Edit
Help

a. Enable heat transfer by activating the energy equation.

\bullet initiality $\rightarrow \equiv$ energy \rightarrow east.	Models -	→ 📑	Energy	→ Edit.
--	----------	-----	--------	---------

💶 Energy	-X
Energy	
👿 Energy Equation	
OK Cancel	Help

Note

You can also double-click a list item in order to open the corresponding dialog box.

- i. Enable the **Energy Equation** option.
- ii. Click **OK** to close the **Energy** dialog box.
- b. Enable the k- ε turbulence model.

 $\mathbf{O} \mathsf{Models} \to \mathbf{F} \mathsf{Viscous} \to \mathsf{Edit...}$

E Viscous Model	
Viscous Model Nodel 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) k-epsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Scalable Wall Functions Enhanced Wall Treatment User-Defined Wall Functions Fnhanced Wall Treatment Options Pressure Gradient Effects Thermal Effects Options	Model Constants Cmu 0.09 C1-Epsilon = 1.44 = C2-Epsilon = I.92 TKE Prandtl Number I • User-Defined Functions • Turbulent Viscosity • none • TKE Prandtl Number • Inone • TDR Prandtl Number • Inone • IDR Prandtl Number • Inone • IDR Prandtl Number • Inone • IDR Prandtl Number • IDR Prandtl Number • Inone •
Options Viscous Heating Curvature Correction Production Kato-Launder Production Limiter OK	Cancel Help

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

Note

The Viscous Model dialog box will expand.

ii. Use the default **Standard** from the **k-epsilon Model** list.

iii. 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 singlephase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model refer to Setting Up the k- ε Model in the User's Guide.

- iv. Click OK to accept the model and close the Viscous Model dialog box.
- 4. Set up your materials for the CFD simulation.

Ŷ	M	ate	eria	als

Materials
Materials
Fluid air Solid aluminum
Create/Edit Delete
Help

a. Create a new material called **water** using the **Create/Edit Materials** dialog box (Figure 1.14: The Create/Edit Materials Dialog Box (p. 36)).

- i. Type water for Name.
- ii. Enter the following values in the **Properties** group box:

Property	Value
Density	1000 kg/m^3
c_p (Specific Heat)	4216 <i>J/kg-K</i>

Property	Value
Thermal Conductivity	0.677 <i>W/m-K</i>
Viscosity	8e-04 <i>kg/m-s</i>

Figure 1.14: The Create/Edit Materials Dialog Box

lame	Material Type		Order Materials by
water	fluid		O Name
Chemical Formula	Fluent Fluid Materials		Chemical Formula
	water		Fluent Database
	Mixture		User-Defined Database.
	none		-
roperties			
Density (kg/m3)	onstant	Edit	
	1000		
Cp (Specific Heat) (j/kg-k)			
		Edit	
	4216		
Thermal Conductivity (w/m-k)	onstant		
	0.677		
Viscosity (kg/m-s)	onstant	Edit	
	0.0008		
		-	

iii. Click Change/Create.

Note

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

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

Note

Both the materials will also be listed under Fluid in the Materials task page.

- v. Close the Create/Edit Materials dialog box.
- 5. Set up the cell zone conditions for the CFD simulation.



Cell Zone Conditions

Zone		
fluid		
Phase	Туре	ID
mixture	✓ fluid	3
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh		
Porous Formulation Superficial Velo		
 Physical Velocit 		
	/	
Help		

- a. Set the cell zone conditions for the fluid zone.
 - i. Select **fluid** in the **Zone** list in the **Cell Zone Conditions** task page, then click **Edit...** to open the **Fluid** dialog box.

Note

You can also double-click a list item in order to open the corresponding dialog box.

E Fluid	
Zone Name	
fluid	
Material Name water]
Frame Motion Laminar Zone Source Terms	
Mesh Motion Fixed Values Forous Zone	
Reference Frame Mesh Motion Porous Zone Embed	dded LES Reaction Source Terms Fixed Values Multiphase
Rotation-Axis Origin	Rotation-Axis Direction
X (in) 0 constant -	X 0 constant -
Y (in) 0 constant 🗸	Y 0 constant -
Z (in) 0 constant	Z 1 constant -
1	Ŧ
OK	Cancel Help

- ii. In the Fluid dialog box, select water from the Material Name drop-down list.
- iii. Click **OK** to close the **Fluid** dialog box.
- 6. Set up the boundary conditions for the CFD analysis.

Conditions

Zone		
interior-fluid		
pressure-outlet		
symmetry velocity-inlet-large	3	
velocity-inlet-smal		
wall-fluid		
Phase	Туре	ID
mixture	✓ velocity-inlet	7
mixture	velocity-iniet •	/
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh	Periodic Conditions	
Highlight Zone		
Help		

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

$\clubsuit Boundary \ Conditions \rightarrow \overleftarrow{\equiv} velocity-inlet-large \rightarrow Edit...$

This opens the **Velocity Inlet** dialog box.

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 (\nearrow) 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.

Velocity Inlet			
Zone Name			
velocity-inlet-large			
Momentum Thermal Radiation Species	DPM Multiphase UI	os	
Velocity Specification Method	Components		
Reference Frame	Absolute	•	
Supersonic/Initial Gauge Pressure (pascal)	0	constant 🔻	
Coordinate System	Cartesian (X, Y, Z)	•	
X-Velocity (m/s)	0.4	constant 🔻	
Y-Velocity (m/s)	0	constant 👻	
Z-Velocity (m/s)	0	constant 👻	
Turbulence			
Specification Method Intensity and Hydraulic Diameter			
Turbulent Intensity (%) 5			
Hydraulic Diameter (mm) 100			
OK Cancel Help			

i. Select Components from the Velocity Specification Method drop-down list.

Note

The **Velocity Inlet** dialog box will expand.

- ii. Enter 0.4 m/s for **X-Velocity**.
- iii. Retain the default value of 0 m/s for both **Y-Velocity** and **Z-Velocity**.
- iv. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
- 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.

Selocity Inlet	x
Zone Name	
velocity-inlet-large	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Temperature (k) 293.15 constant	
OK Cancel Help	

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

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

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{\equiv}$ velocity-inlet-small \rightarrow Edit...

Velocity Specification Method	Components
X-Velocity	0 <i>m/s</i>
Y-Velocity	1.2 <i>m/s</i>
Z-Velocity	0 <i>m/s</i>
Specification Method	Intensity & Hydraulic Diameter

Velocity Specification Method	Components
Turbulent Intensity	5%
Hydraulic Diameter	25 mm
Temperature	313.15 <i>K</i>

- c. Set the boundary conditions at the outlet (**pressure-outlet**), as shown in the **Pressure Outlet** dialog box.
 - $\textcircled{P} Boundary Conditions \rightarrow \fbox{pressure-outlet} \rightarrow \texttt{Edit...}$

Pressure Outlet
Zone Name
pressure-outlet
Momentum Thermal Radiation Species DPM Multiphase UDS Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method Normal to Boundary
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.

7. Set up solution parameters for the CFD simulation.

Note

In the steps that follow, you will set up and run the calculation using the task pages listed under the **Solution** heading in the navigation pane.

a. Change the Gradient method.

CSolution Methods

Solution Methods			
Pressure-Velocity Coupling			
Scheme			
SIMPLE			
Spatial Discretization			
Gradient	Â.		
Green-Gauss Node Based 🔹			
Pressure			
Second Order 🔹	=		
Momentum	-		
Second Order Upwind 🔹			
Turbulent Kinetic Energy			
First Order Upwind 🔹	ш.		
Turbulent Dissipation Rate			
First Order Upwind 🔹	Ŧ		
Transient Formulation			
· · · · · · · · · · · · · · · · · · ·			
Non-Iterative Time Advancement			
Frozen Flux Formulation Pseudo Transient			
High Order Term Relaxation Options			
Default			
Help			

In the **Spatial Discretization** section of the **Solution Methods** pane, change the **Gradient** to **Green-Gauss Node Based**. This gradient method is suggested for tetrahedral meshes.

b. Examine the convergence criteria for the equation residuals.



Residual Monitors					×
Options Image: Print to Console Image: Print to Console </td <td>Equations Residual continuity x-velocity y-velocity</td> <td>Monitor</td> <td>Check Convergence</td> <td>Absolute Criteria 0.001 0.001 0.001</td> <td></td>	Equations Residual continuity x-velocity y-velocity	Monitor	Check Convergence	Absolute Criteria 0.001 0.001 0.001	
Iterations to Plot	z-velocity			0.001	-
	Residual Values			Convergence C	riterion
Iterations to Store	Normalize		Iterations 5	absolute	•
	Scale				
	Compute Loca	l Scale			
OK Plot Renormalize Cancel Help					

- i. Ensure that **Plot** is enabled in the **Options** group box.
- 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 monitor at the outlet (pressure-outlet)

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

♦ Monitors (Surface Monitors) → Create...

Surface Monitor			
Name surf-mon-1	Report Type Facet Maximum		
Options V Print to Console	Field Variable Temperature		
Vindow	Static Temperature		
2 Curves Axes Write File Name F:/H1_tutorial_re-runs/Tutorial 1/elbow-w	interior-fluid pressure-outlet symmetry velocity-inlet-large velocity-inlet-small wall-fluid		
X Axis Iteration Get Data Every 3 Iteration V	Waii-Iiuiu		
Average Over(Iterations)	☐ Highlight Surfaces New Surface ▼		
OK Cancel Help			

- i. Retain the default entry of **surf-mon-1** for the **Name** of the surface monitor.
- ii. Enable the **Plot** option for **surf-mon-1**.
- iii. Set Get Data Every to 3 by clicking the up-arrow button.

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

- iv. Select Facet Maximum from the Report Type drop-down list.
- v. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- vi. Select pressure-outlet from the Surfaces selection list.
- vii. Click **OK** to save the surface monitor settings and close the **Surface Monitor** dialog box.

The name and report type of the surface monitor you created will be displayed in the **Surface Mon-***itors* selection list in the **Monitors** task page.

d. Initialize the flow field.

Solution Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
More Settings Initialize
Reset DPM Sources Reset Statistics
Help

- i. Keep the default of **Hybrid Initialization** from the **Initialization Methods** group box.
- ii. Click Initialize.
- e. Check to see if the case conforms to best practices.

\mathbf{Q} Run Calculation \rightarrow Check Case

Case Check	×
Mesh Models Boundaries and Cell Zones Materials Solver	
Manual Implementation	
Recommendation	-
Consider using higher order discretization for improved accuracy of the final solution. First order discretization may be used in the initial solution. (Solution Methods)	
	~
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.
- 8. Calculate a solution.
 - a. Start the calculation by requesting 300 iterations.

CRun Calculation

Run Calculation
Check Case Preview Mesh Motion
Number of Iterations Reporting Interval
Profile Update Interval
Data File Quantities Acoustic Signals
Calculate
Help

- i. Enter 300 for Number of Iterations.
- ii. Click Calculate.

Important

Note that the ANSYS Fluent settings file (FFF.set) is updated before the calculation begins.

Important

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).

Note

As the calculation progresses, the surface monitor history will be plotted in the graphics window (Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet (p. 49)).

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.

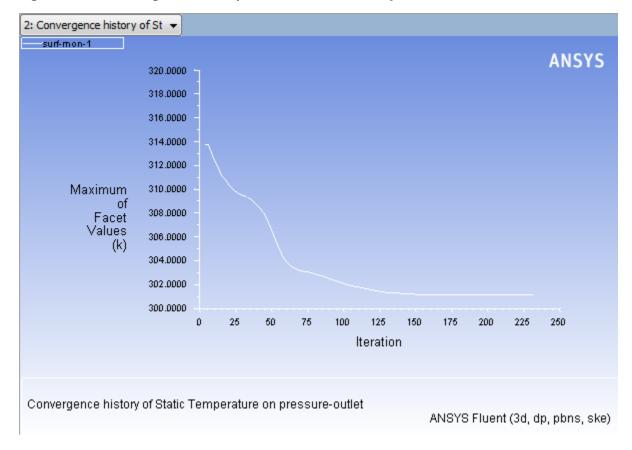


Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet

You can display the residuals history (Figure 1.16: Residuals for the Converged Solution (p. 50)), by selecting it from the graphics window drop-down list.

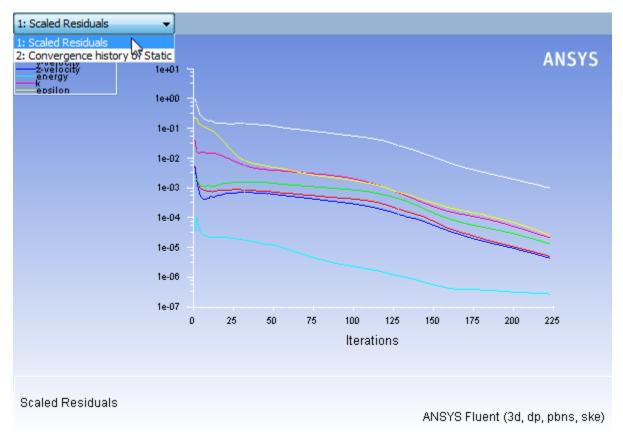


Figure 1.16: Residuals for the Converged Solution

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.

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

```
View \rightarrow Files
```

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

▼	A	
1	🔄 Fluid Flow (Fluent)	
2	🕅 Geometry	× 🖌
3	🎯 Mesh	× 🖌
4	🍓 Setup	× 🖌
5	Solution	× 🖌
6	🥩 Results	2
elbow		

1.4.6. 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, then review the list of files generated by ANSYS Workbench.

1. Display results in ANSYS Fluent.

With ANSYS Fluent still running, 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.

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



Note

You can also double-click a list item in order to open the corresponding dialog box.

Contours		_		
Options	Contours of			
Filled	Velocity	•		
Node Values	Velocity Magnitude			
Global Range	Velocity Magnitude	•		
Auto Range	Min (m/s) Max (m/s)			
Clip to Range	0 1.364282			
Draw Mesh	Surfaces			
E Bran Heart	interior-fluid			
	pressure-outlet	Â		
Levels Setup	symmetry	=		
20 🚔 1	velocity-inlet-large			
	velocity-inlet-small	-		
Surface Name Pattern				
	New Surface 💌			
Match	Surface Types			
	axis	*		
	clip-surf			
	exhaust-fan			
	fan	-		
Display Compute Close Help				

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

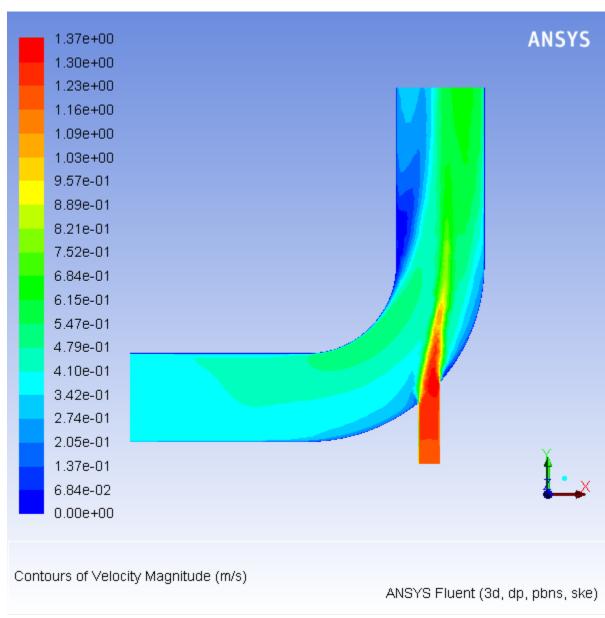


Figure 1.17: Velocity Distribution Along Symmetry Plane

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

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

Contours		
Options	Contours of	
V Filled	Temperature	-
V Node Values	Static Temperature	
 Global Range Auto Range 	Min (k) Max (k)	
Clip to Range	292.9372 313.2289	
Draw Mesh	Surfaces	
	interior-fluid	*
Levels Setup	pressure-outlet	
	symmetry	E
20 🔺 1	velocity-inlet-large	
	velocity-inlet-small	Ŧ
Surface Name Pattern	New Surface	
Ma	Surface Types	
	axis	*
	clip-surf	
	exhaust-fan	
	fan	-
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.

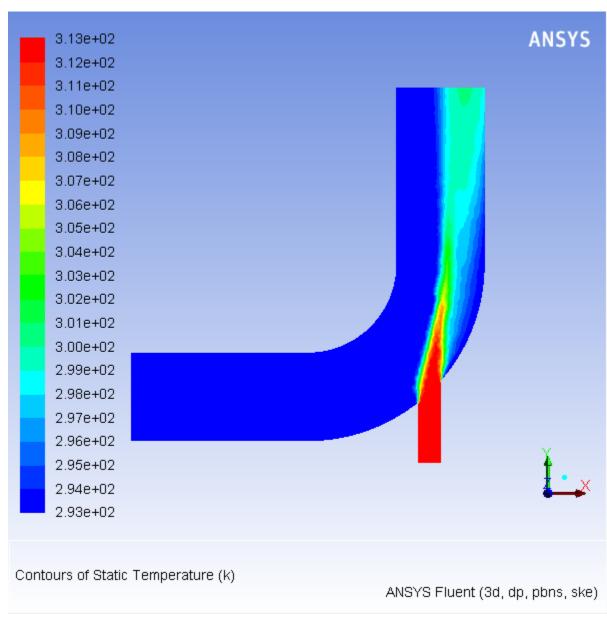


Figure 1.18: Temperature Distribution Along Symmetry Plane

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.

 $View \rightarrow Files$

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.

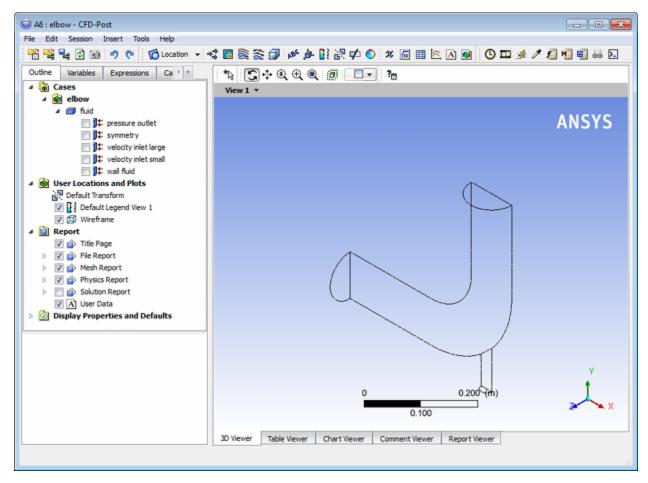
- 2. Display results in CFD-Post.
 - a. Start CFD-Post.

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.

Figure 1.19: The Elbow Geometry 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* () is disabled.
- d. Display filled 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.

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

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, select fluid in the Domains list.
- iv. Select symmetry in the Locations list.
- v. Select Velocity in the Variable list.
- vi. Click **Apply**.

				_
Details of Conto	ur 1			
Geometry I	abels	Render	View	_
Domains	fluid		•	1
Locations	symmet	ry	•	
Variable	Velocity	/	• E	
Range	Global		•	
Min			0 [m s^-1]	
Max			1.36322 [m s^-1]	
Boundary Data	C	Hybrid	Onservative	
Apply			Reset Defaults	

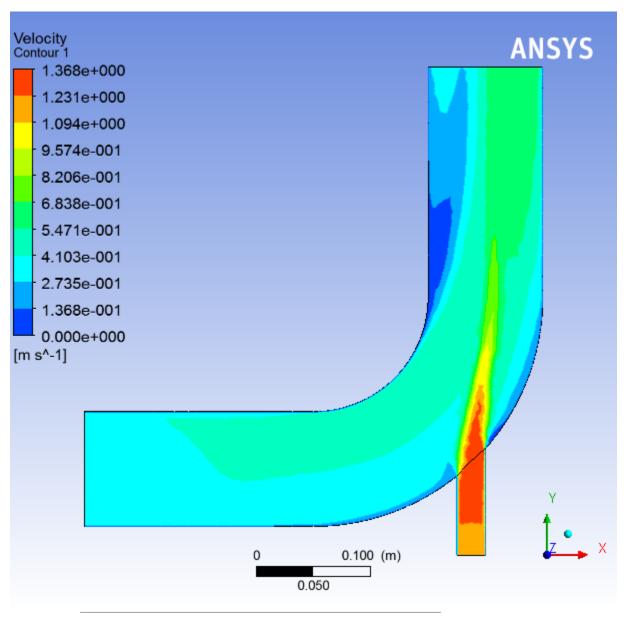


Figure 1.20: Velocity Distribution Along Symmetry Plane

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

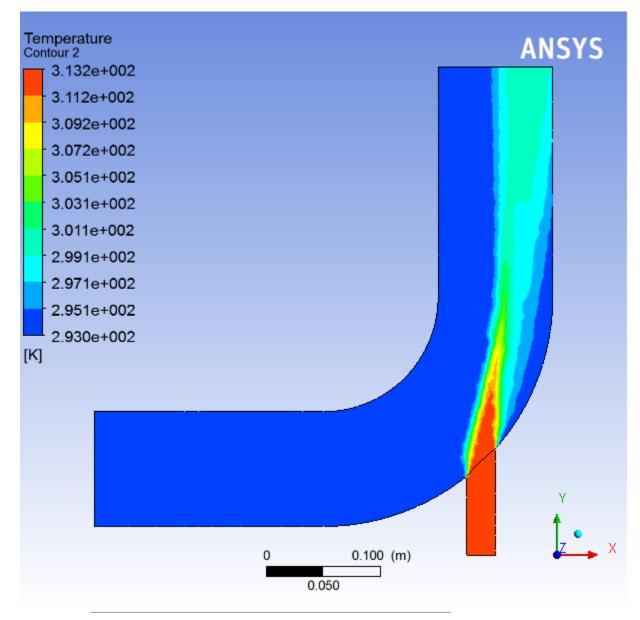
$\textbf{Insert} \rightarrow \textbf{Contour}$

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, select **fluid** from the **Domains** list.

- v. Select symmetry in the Locations list.
- vi. Select Temperature in the Variable list.
- vii. Click **Apply**.





3. Close the CFD-Post application by selecting **File** → **Close ANSYS 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.

$View \rightarrow Files$

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.

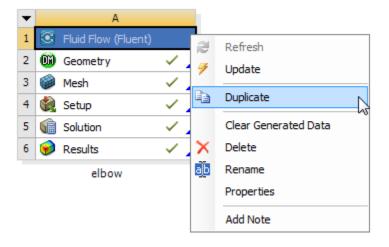
1.4.7. Duplicating the Fluent-Based Fluid Flow Analysis System

At this point, you have a completely defined fluid flow system that is comprised of 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



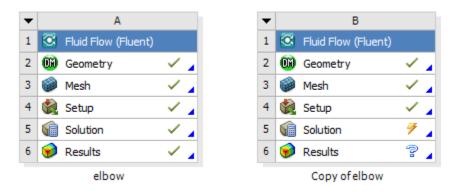


Figure 1.23: The Original Fluid Flow System and Its Duplicate

Note

Notice that in the duplicated system, the state of the **Solution** cell indicates that the cell requires an update 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.

1.4.8. 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.

Double-click the **Geometry** cell of the new-elbow system (cell B2) to display the geometry in ANSYS DesignModeler.

- 2. Change the diameter of the small inlet (velocity-inlet-small).
 - a. 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.

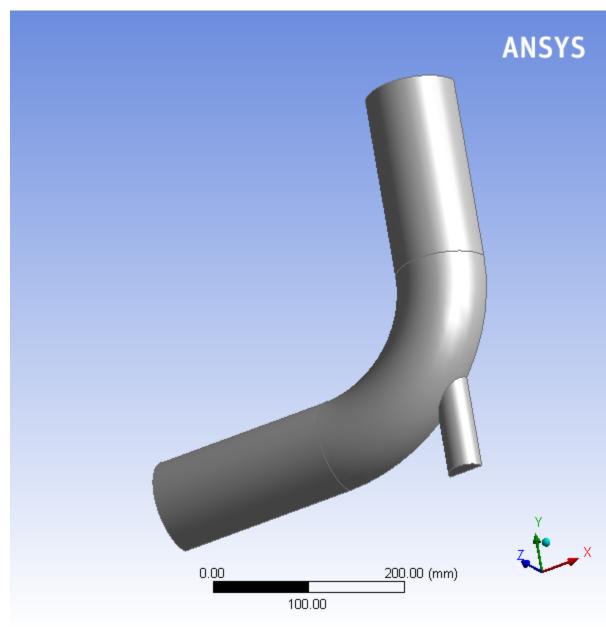


Figure 1.24: Changing the Diameter of the Small Inlet in ANSYS DesignModeler

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

$View \rightarrow Files$

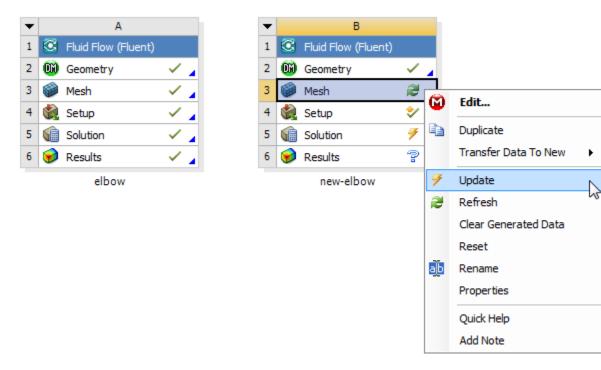
Note the addition of the geometry, mesh, and ANSYS Fluent settings files now associated with the new, duplicated system.

1.4.9. 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 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.

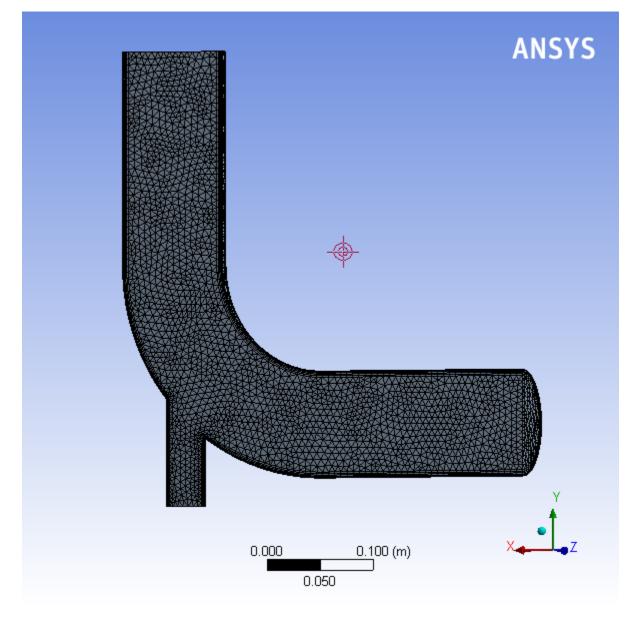


Figure 1.26: The Updated Geometry and Mesh in the ANSYS Meshing Application

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

$View \rightarrow Files$

1.4.10. Calculating a New Solution in ANSYS Fluent

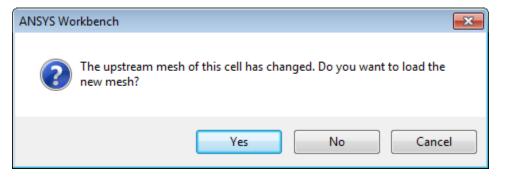
Now that there is an updated computational mesh for the modified geometry in the duplicated system, a new solution must be generated using ANSYS Fluent. In this step, you will revisit the settings within ANSYS Fluent, calculate another solution, view the new results, then review the list of files generated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

Figure 1.27: ANSYS Workbench Prompt When the Upstream Mesh Has Changed



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

\$General → Units...

3. Check the mesh (optional).

\bigcirc General \rightarrow Check

4. Revisit the boundary conditions for the small inlet.

Here, you must set the hydraulic diameter to 38 mm based on the new dimensions of the small inlet.

5. Re-initialize the solution.

Solution Initialization

Keep the default Hybrid Initialization and click Initialize.

6. Recalculate the solution.

Run Calculation

Keep the Number 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.

1.4.11. 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.

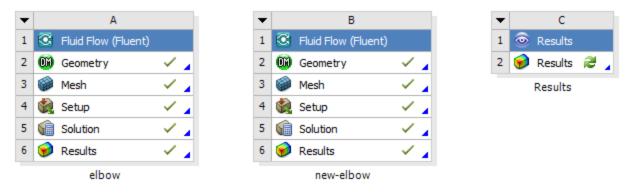
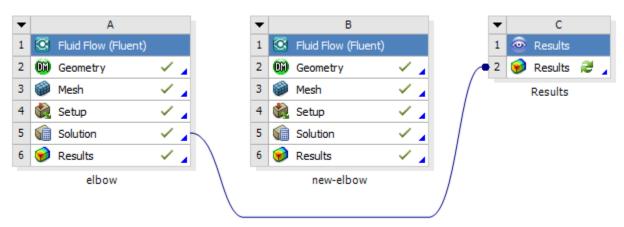


Figure 1.28: The NewResults System in the Project Schematic

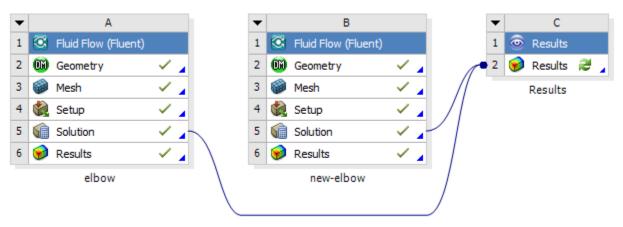
- 2. Add the solutions of each of the systems to the **Results** system.
 - a. Select the **Solution** cell in the first Fluid Flow analysis system (cell A5) and drag it over the **Results** cell in the **Results** system (cell C2). This creates a transfer data connection between the two systems.

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



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.

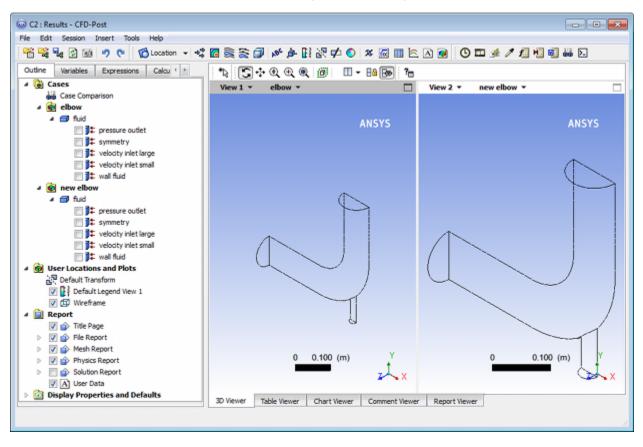




3. Open CFD-Post to compare the results of the two fluid flow systems.

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.

Figure 1.31: CFD-Post with Both Fluid Flow Systems Displayed



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

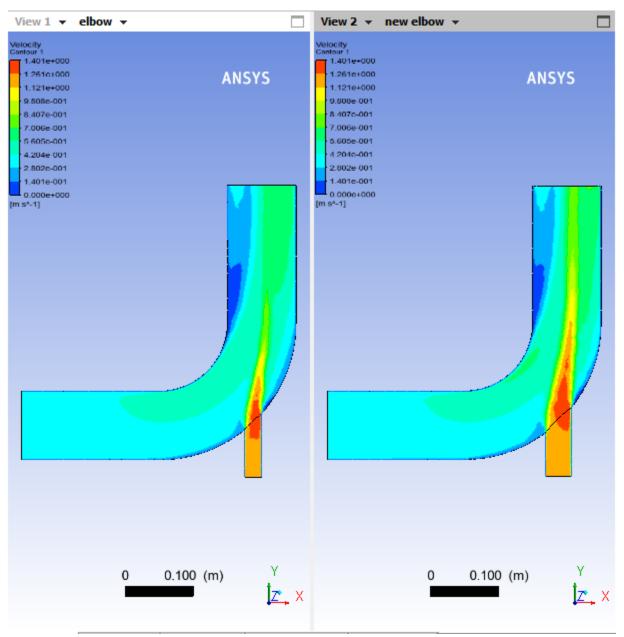
Alternatively, you can select the synchronization tool (E) in the **3D Viewer Toolbar** to synchronize the views, so that when you re-orient one view, the other view is automatically updated.

- b. Display filled 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, select **fluid** in the **Domains** list.
- iv. Select symmetry in the Locations list.
- v. Select Velocity in the Variable list.
- vi. Click **Apply**. The velocity contours are displayed in each view.





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

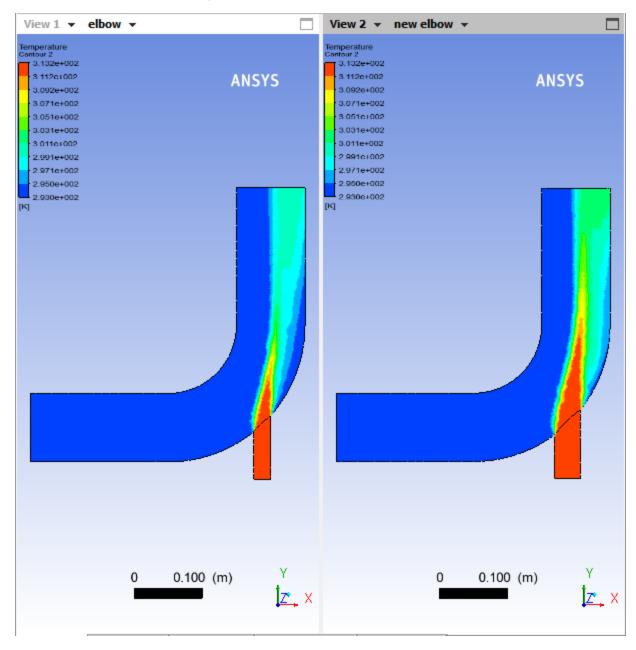
$\textbf{Insert} \rightarrow \textbf{Contour}$

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 1.33: 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 etc. 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, etc., 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.

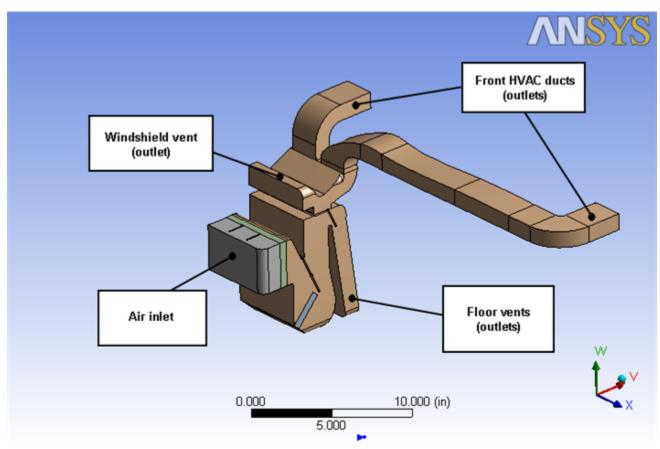
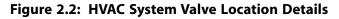


Figure 2.1: Automotive HVAC System



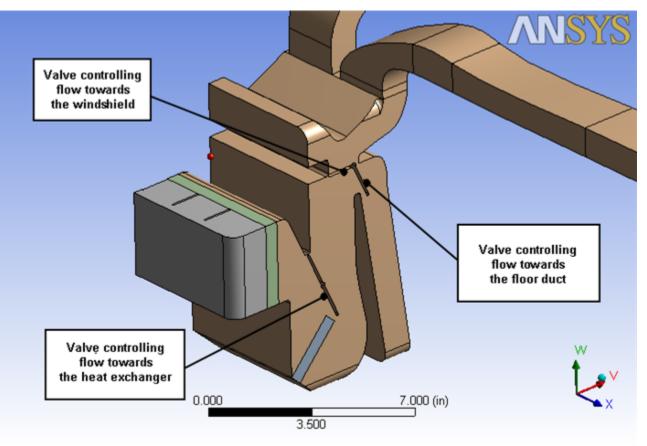


Figure 2.1: Automotive HVAC System (p. 75) shows a representative automotive HVAC system. The system has three valves (as shown in Figure 2.2: HVAC System Valve Location Details (p. 76)), which control the flow in the HVAC system. The three valves control:

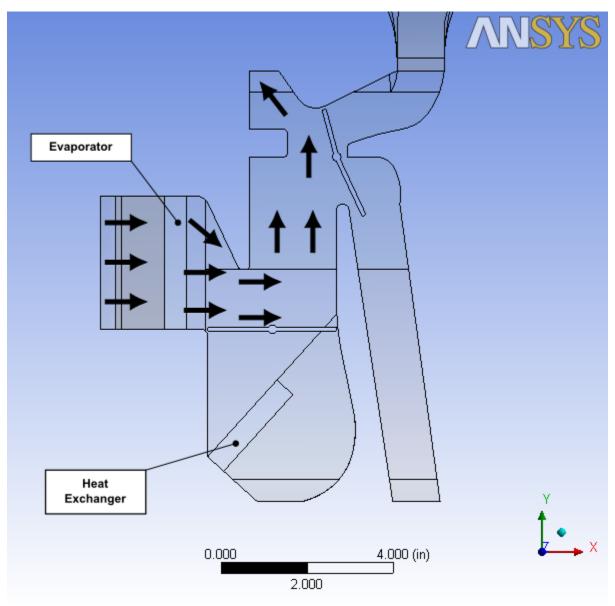
- Flow over the heat exchanger coils
- Flow towards the duct controlling the flow through the floor vents
- · Flow towards the front vents or towards the windshield

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

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 and Running the Simulation
- 2.4.5. Postprocessing and Setting the Output Parameters in ANSYS CFD-Post
- 2.4.6. Creating Additional Design Points in ANSYS Workbench
- 2.4.7. Postprocessing the New Design Points in CFD-Post
- 2.4.8. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip the workbench-parameter-tutorial_R150.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. Start ANSYS Workbench by clicking the Windows **Start** menu, then selecting the **Workbench** option in the **ANSYS 15.0** program group.

Start → All Programs → ANSYS 15.0 → Workbench 15.0

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, fluentworkbench-param.wbpj, and a corresponding project folder, fluent-workbenchparam_files, for this tutorial.

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



A fluent-workbench-param - Workbench		
File View Tools Units Extensions H	telp	
Project		
Import 🖓 Reconnect 🔮 Refresh Projec	ct 🍠 Update Project 🧚 Update All Design Points	
	Project Schematic	- 4 X
Analysis Systems		
Fluid Flow - Blow Molding (Polyflow)		
Fluid Flow-Extrusion(Polyflow)	▼ A	
Fluid Flow (Fluent)	1 🕄 Fluid Flow (FLUENT)	
Fluid Flow (Polyflow)	2 🔞 Geometry	
IC Engine	3 Mesh	
C Throughflow		
E Component Systems	4 🍓 Setup 👕 🖌	
😹 BladeGen	5 🍿 Solution 👕 🥊	
External Data	6 🎯 Results 🛛 🚏 🖌	
External Model	→ 7 🛱 Parameters	
Finite Element Modeler	Child Charles (Children)	
E Fluent	Fluid Flow (FLUENT)	
Fluent (with TGrid meshing)		
Geometry		
Mesh		
Microsoft OfficeExcel	Parameter Set	
22 Polyflow		
Polyflow - Blow Molding		
Polyflow - Extrusion		
Results		
System Coupling		
Vista AFD		
Vista CCD		
Vista CCD (with CCM)		
Vista CPD		
Vista RTD		
yista TF		
Design Exploration		
External Connection Systems		
View Al / Customize		
🔋 Ready		Show 4 Messages

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.

 $View \rightarrow Files$

🔥 fluent-workbench-param - Workbench						
File View Tools Units Extensions Help						
🎦 💕 🛃 🔣 📄 Project						
Import	t 🗲 Up	date Project 💔 Update All Design Points				
Toolbox - 9 X	Project 1	Schematic				- џ X
Analysis Systems						~
S Fluid Flow - Blow Molding (Polyflow)						
Fluid Flow-Extrusion(Polyflow)		▼ A				
Fluid Flow (Fluent)		1 C Fluid Flow (FLUENT)				
Fluid Flow (Polyflow)		2 🚯 Geometry				
IC Engine		3 📦 Mesh 🖉				E
C Throughflow						
Component Systems		4 🍓 Setup 🛛 🖓 🖌				
💏 BladeGen		5 🕼 Solution 👕 🖌				
External Data		6 🥩 Results 🛛 🚏 🖌				
External Model	\rightarrow	7 Darameters				
Finite Element Modeler		Fluid Flow (FLUENT)				
Fluent		FILID FILW (FLOENT)				
Fluent (with TGrid meshing)						-
Geometry						
i Mesh	Files					. ģ X
Microsoft OfficeExcel		A	в	с	D	
22 Polyflow	1	Name	Cel 🕞	Size 💌	Type 💌	Date I
Polyflow - Blow Molding	1	Name 🖸	ID 🛄	Size 💽	Туре 💌	Dater
12 Polyflow - Extrusion	2	A fluent-workbench-param.wbpj		134 KB	Workbench Project	1/7/2013 3
Results	-	•			Fle	
System Coupling	3	🥪 Geom.agdb	A2	2 MB	Geometry File	1/7/2013 3
Vista AFD		-				
Vista CCD	4	🥪 FFF.agdb	A2	2 MB	Geometry File	1/7/2013 3
Vista CCD (with CCM)						
Vista CPD	5	FFF.mshdb	A3	184 KB	.mshdb	1/7/2013 3
Vista RTD		_				
Vista TF	6	k designPoint.wbdp		37 KB	Workbench Design	1/7/2013 3
Design Exploration					Point File	411120200
External Connection Systems	7 DesignPointLog.csv 2KB .csv 1/			1/7/2013 3		
View All / Customize	•					÷
Ready				Show F	Progress 🛛 🤛 Show 4 Me	ssages:

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 ([?]). For more information about cell states, see Understanding Cell States.

- 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	Гр Р1	hcpos	90	
5	Ф Р2	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. Create three new input parameters.

In the row that contains **New input parameter**, click the parameter table cell under the **Parameter Name** column (the cell with **New name**) to create a new named input parameter. Create three new parameters named input_hcpos, input_ftpos, and input_wsfpos. Note the ID of the parameter that appears in column A of the table. For the new input parameters, the parameter IDs would be **P4**, **P5**, and **P6**, respectively. In the **Values** column, enter values for each new parameter of 15, 25, and 90, respectively.

	А	В	с	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	🗏 🙆 Fluid Flow (FLUENT) (A1)			
4	🗘 P1	hcpos	90	
5	ί <mark>ρ</mark> Ρ2	ftpos	25	
6	🗘 РЗ	wsfpos	175	
7	ф Р4	input_hcpos	15	
8	ф Р5	input_ftpos	25	
9	ф 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 having the parameter ID of 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.

Figure 2.8:	Constrained	Parameter hcpos
-------------	-------------	------------------------

Outline o	Outline of All Parameters 🔹 🔻 🗙				
	A		В	с	
1	ID		Parameter Name	Value	
2	Input Parameters				
3	🖃 🙆 Fluid Flow (FLU	ENT) (A1)			
4	ί <mark>ρ</mark> Ρ1		hcpos	25	
5	ί <mark>ρ</mark> Ρ2		ftpos	25	
6	ι <mark>φ</mark> P3		wsfpos	175	
7	🛱 P4		input_hcpos	15	
8	🛱 P5		input_ftpos	25	
9	ф Р6		input_wsfpos	90	
*	🗘 New input para	meter	New name	New expression	
11	 Output Parameters 				
*	New output parameter			New expression	
13	Charts				
•		111		•	
Propertie	es of Outline A4: P1			⊸ џ х	
	А		В		
1	Property		Value		
2	■ General				
3	Expression min(max(25,P4),90)				
4	Description				
5	Error Message				
6	Expression Type	Derived			
7	Usage	Input			
· ·	_				

7. 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 o	of All Parameters			▼ Ŧ X
	A		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	🛱 P2		ftpos	25
6	🗘 P3		wsfpos	175
7	ι <mark>φ</mark> Ρ4		input_hcpos	15
8	ι <mark>φ</mark> Ρ5		input_ftpos	25
9	ф Рб		input_wsfpos	90
*	l New input parameter		New name	New expression
11	Output Parameters			
*	New output parameter			New expression
13	Charts			
•		111		•
Propertie	es of Outline B5: P2			- † X
	A		В	
1	Property		Value	
2	General			
3	Expression min(max(2		0,P5),60)	
4	Description			
5	Error Message			
6	Expression Type	Derived		
7	Usage	Input		
		Dimensionless		

Figure 2.9: Constrained Parameterftpos

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

Figure 2.10:	Constrained	Parameter wsfpos
--------------	-------------	-------------------------

Outline of All Parameters 🔹 🗣 🗘				
	A		В	с
1	ID		Parameter Name	Value
2	Input Parameters			
3	🖃 🙆 Fluid Flow (FLU	ENT) (A1)		
4	<mark>្រុំ</mark> P1		hcpos	25
5	ι <mark>ρ</mark> Ρ2		ftpos	25
6	ι <mark>ρ</mark> Ρ3		wsfpos	90
7	ί <mark>ρ</mark> Ρ4		input_hcpos	15
8	ι <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 parameter			New expression
13	Charts			
•	m			- F
Properti	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	pression 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 → 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.

1. Start 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

I Fluent Launcher (Setting Edit Only)	
ANSYS	Fluent Launcher
Dimension 2D 3D	Options Double Precision Meshing Mode
Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme Do not show this panel again Show More Options	Processing Options Serial Parallel
	ancel <u>H</u> elp 🔻

a. 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.

i. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, 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.

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

Important

The memory requirements of this tutorial exceed the 2 GB per process limit of 32–bit Windows platforms. If you are planning to run this tutorial on a 32–bit Windows platform, it is necessary to enable parallel processing by selecting **Parallel** under **Processing Options** and setting **Number of Processes** to at least 2. Note that you must have the correct license support in order to use parallel processing.

Parallel processing also offers a substantial reduction in computational time. Refer to Parallel Processing (p. 1129) in this manual and Starting Parallel ANSYS Fluent Using Fluent Launcher in the User's Guide for further information about using the parallel processing capabilities of ANSYS Fluent.

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

Note

Fluent will retain your preferences for future sessions.

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

	Fluent [3d, pbms, lam] [ANSYS CFD] olve Adapt Surface Display Report Parallel 1	feer Hels
	S+QQ / Q, M.T. II.	ne nep
Meshing	General	2 Medh •
Mashing Meth Granation Solution Select Models Models Phase Call Zone Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Solution (Network) Solution California Calculation Activities Ruin Calculation Calculation Activities Ruin Calculation Resalts Calculation Activities Ruin Calculation Resalts Calculation Activities Ruin Calculation Resalts	Mech Scale Check Report Quality Display Solver Type Pressure-Sased Danaity-Sased Relative Tana Stady Tanaient Caranty Ubria	ANSYS
		Mesh
		ANSYS Fluert (dd, phons, lam) writing wall-inlet-flow-diverter (type wall) (nixture) Done. writing wall-evaparator (type wall) (nixture) Done. writing vall-entral-unix (type wall) (nixture) Done. writing symmetry-entral-unix (type symmetry) (nixture) Done. writing symmetry-entral-unix (type symmetry) (nixture) Done. writing symmetry-entral-unix (type symmetry) (nixture) Done. writing symmetry-entral-entral-unix flype symmetry (type listerior) (nixture) Done. writing interior-fluid-central-unit-fluid-evaparator (type interior) (nixture) Done. writing interior-fluid-central-unit-fluid-evaparator (type interior) (nixture) Done. writing zones nap name-id Done.

Figure 2.12: The ANSYS Fluent Application

2. Reorder the mesh.

$\textbf{Mesh} \rightarrow \textbf{Reorder} \rightarrow \textbf{Domain}$

This is done to reduce the bandwidth of the cell neighbor number and to speed up the computations. This is especially important for large cases involving 1 million or more cells. The method used to reorder the domain is the Reverse Cuthill-McKee method.

- 3. Set up your models for the CFD simulation.
 - a. Enable heat transfer by activating the energy equation.

• Models → = Energy → Edit				
💶 Energy 🛛 💽				
Energy				
OK Cancel Help				

- i. Enable the **Energy Equation** option.
- ii. Click **OK** to close the **Energy** dialog box.
- b. Enable the k- ε turbulence model.

 \bigcirc Models $\rightarrow \stackrel{\frown}{\equiv}$ Viscous \rightarrow Edit... Viscous Model х Model Model Constants Inviscid Cmu Laminar 0.09 Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) C1-Epsilon k-omega (2 eqn) 1.44 Transition k-kl-omega (3 eqn) Transition SST (4 eqn) C2-Epsilon Reynolds Stress (7 eqn) 1.92 Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) TKE Prandtl Number Large Eddy Simulation (LES) 1 k-epsilon Model User-Defined Functions Standard RNG Turbulent Viscosity Realizable none • Near-Wall Treatment Prandtl Numbers Standard Wall Functions TKE Prandtl Number Scalable Wall Functions none Ŧ Non-Equilibrium Wall Functions Ξ TDR Prandtl Number Enhanced Wall Treatment O User-Defined Wall Functions none Ŧ Energy Prandtl Number Enhanced Wall Treatment Options none Ŧ Pressure Gradient Effects Thermal Effects Options Viscous Heating Curvature Correction Production Kato-Launder Production Limiter OK Cancel Help

i. Select k-epsilon (2 eqn) from the Model list.

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 Wall Treatment for RANS Models in the User's Guide.

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

2.4.4. Defining Input Parameters in ANSYS Fluent and Running the Simulation

You have now started setting up the CFD analysis using ANSYS Fluent. In this step, you will define input parameters for the velocity inlet, define heat source boundary conditions for the evaporator, then calculate a solution.

1. Define an input parameter called in_velocity for the velocity at the inlet boundary.



This displays the **Velocity Inlet** dialog box.

Velocity Inlet	×						
Zone Name							
inlet-air							
Momentum Thermal Radiation Species DPM Multiphase UDS							
Velocity Specification Method Magnitude, Normal to Boundary							
Reference Frame Absolute							
Velocity Magnitude (m/s)	in_velocity						
Supersonic/Initial Gauge Pressure (pascal)	constant 💌						
Turbulence							
Specification Method Intensity and Hydraulic Diameter							
Turbulent Intensity (%) 5 P						
Hydraulic Diameter (m) 0.061						
OK Cancel Help							

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

This displays the Input Parameter Properties dialog box.

💶 Input Parameter Properties 🛛 🛛 📧
Name
in_velocity
Current Value (m/s)
0.5
used In:
OK Cancel Help

- b. In the **Input Parameter Properties** dialog box, enter in_velocity for the **Name**, and enter 0.5 for the **Current Value**.
- c. Click OK to close the Input Parameter Properties dialog box.
- d. In the **Velocity Inlet** dialog box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.
- e. Retain the value of 5 % for **Turbulent Intensity**.
- f. 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 UDS	
	1
Temperature (k) 310 in_temp -	
2	
OK Cancel Help	

- a. In the **Thermal** tab of the **Velocity Inlet** dialog box, select **New Input Parameter...** from the drop-down list for the **Temperature**.
- b. In the **Input Parameter Properties** dialog box, enter in_temp for the **Name**, and enter 310 for the **Current Value**.
- c. Click OK to close the Input Parameter Properties dialog box.
- d. Click **OK** to close the **Velocity Inlet** dialog box.
- 3. Set the turbulence parameters for backflow at the front outlets and foot outlets.

♀Boundary	Conditions \rightarrow	 outlet-front-mid →	Edit

Pressure Outlet
Zone Name
outlet-front-mid
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method Normal to Boundary
Radial Equilibrium Pressure Distribution
C Average Pressure Specification
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Hydraulic Diameter
Backflow Turbulent Intensity (%) 5
Backflow Hydraulic Diameter (m) 0.044
OK Cancel Help

- a. In the **Pressure Outlet** dialog box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.
- b. Retain the value of 5 for Backflow Turbulent Intensity (%).
- c. 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.

- d. Click OK to close the Pressure Outlet dialog box.
- e. Copy the boundary conditions from outlet-front-mid to the other front outlet.

Generations → Copy...

Copy Conditions		×
From Boundary Zone Interior-fluid-central-unit Interior-fluid-central-unit-fluid-evaporator Interior-fluid-central-unit-fluid-heat-exchanger Interior-fluid-evaporator Interior-fluid-heat-exchanger outlet-foot-left <u>outlet-front-mid</u> outlet-front-side-left outlet-windshield	4	To Boundary Zones 🖹 🗐 🗐 outlet-foot-left outlet-front-side-left outlet-windshield
Copy Close Help		

i. Select outlet-front-mid in the From Boundary Zone selection list.

Scroll down, if necessary, to find **outlet-front-mid**.

- 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. Click Close to close the Copy Conditions dialog box.
- f. 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	

g. To see all of the input and output parameters that you have defined in ANSYS Fluent, in the **Boundary onditions** task page, click the **Parameters...** button to open the **Parameters** dialog box.

Parameters	—
Input Parameters	Output Parameters
View Delete More -	Create View More
Close	Help

Figure 2.13: The Parameters Dialog Box 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	<mark>ြို</mark> ့ P1	hcpos	25	
5	ι <mark>φ</mark> Ρ2	ftpos	25	
6	ြို့ P3	wsfpos	90	
7	ြို့ P7	in_velocity	0.5	m s^-1 💌
8	(<mark>p</mark> P9	in_temp	310	к 🔹
9	ί <mark>ρ</mark> Ρ4	input_hcpos	15	
10	ί <mark>ρ</mark> Ρ5	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 Workbench

4. Define a heat source boundary condition for the evaporator volume.

 $\textcircled{Cell Zone Conditions} \rightarrow \overleftarrow{\sqsubseteq} fluid-evaporator \rightarrow Edit...$

💶 Fluid
Zone Name
fluid-evaporator
Material Name air Edit
Frame Motion Laminar Zone V Source Terms
Mesh Motion Fixed Values Forous Zone
Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Multiphase
Z Momentum 0 sources
Turbulent Kinetic Energy 0 sources Edit
Turbulent Dissipation Rate 0 sources
Energy 1 source Edit
,
OK Cancel Help

- a. In the Fluid dialog box, enable Source Terms.
- b. In the **Source Terms** tab, scroll down to **Energy**, and click the **Edit...** button.

🛂 Energy s	ources		x
		Number of Energy sources	
1. (w/m3)	-787401.6	⊂onstant ▼	*
			~
	OK C	ancel Help	

- c. In the Energy sources dialog box, change the Number of Energy sources to 1.
- d. For the new energy source, select **constant** from the drop-down list, and enter -787401.6 W/m³ based on the evaporator load (200 W) divided by the evaporator volume (0.000254 m³) that was computed earlier.
- e. Click OK to close the Energy Source dialog box.
- f. Click **OK** to close the **Fluid** dialog box.
- 5. Set the Solution Methods

CSolution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	•
Least Squares Cell Based 🔹	l
Pressure	l
PRESTO!	
Momentum	-
First Order Upwind 🔹	l
Turbulent Kinetic Energy	l
First Order Upwind 🔹	
Turbulent Dissipation Rate	
First Order Upwind 🔹 🗸	÷
Transient Formulation	
	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
High Order Term Relaxation Options	
Default	
Help	

a. Select **Coupled** in the **Scheme** drop-down list.

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

b. Select **PRESTO!** for **Pressure** and **First Order Upwind** for **Momentum** and **Energy** in the **Spatial Discretization** group box.

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.

6. Initialize the flow field.

Solution Initialization

Solution Initialization			
Initialization Methods Hybrid Initialization Standard Initialization			
More Settings Initialize Patch Reset DPM Sources Reset Statistics			
Help			

- a. Retain the default selection of **Hybrid Initialization**.
- b. Click the **Initialize** button.
- 7. Run the simulation in ANSYS Fluent.

CRun Calculation

Run Calculation
Check Case Preview Mesh Motion
Number of Iterations Reporting Interval
Profile Update Interval
Data File Quantities Acoustic Signals
Calculate
Help

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

The solution converges within approximately 60 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 solve/set/flow-warnings text user interface (TUI) command to no in the console.

8. Close Fluent.

File → Close Fluent

9. Save the project in ANSYS Workbench.

File \rightarrow Save

2.4.5. 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.

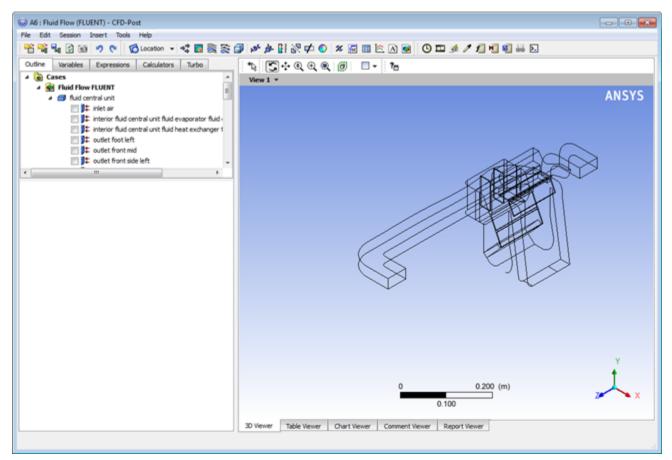


Figure 2.15: The Automotive HVAC Geometry Loaded into CFD-Post

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

$\mathbf{Edit} \rightarrow \mathbf{Options...}$

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

Object Highlighting			
Туре	Surface Mesh 👻		
Pre-genera	ate region highlight		
Background			
Mode	Color 🗸		
Color Type	Solid 🗸		
Color			
ANSYS Logo	Black		
Text Color			
Edge Color			
 Axis Visibility Ruler Visibility 			
Stereo			
Mode	Normal 👻		
Stereo Effect	Weaker Stronger		
Note: The stereo Mode setting only takes effect the next time you run the application. For stereo viewing to work, you need to turn on 'Stereo' in your graphic card, have a display that supports stereo, and ensure that your view is set to Perspective mode (Right-dick in viewer > Projection > Perspective).			

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



You can also click the ellipsis icon <u></u>to bring up a color selector dialog box from 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. Double-click **Default Legend View 1** in the tree view to display the **Details** view for the default legend to be used for your plots.
 - b. In the **Details** view for **Default Legend View 1**, in the **Definition** tab, change the **Title Mode** to **Variable**.

Details of Default Legend View 1								
Definition	Appearance							
Title Mode	Variable							
Show Legend Units								
Vertical	Horizontal							
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 Parameters								
Size	0.6							
Aspect	0.07							
Text Paramete	ers							
Precision	2 Fixed V							
Value Ticks	5							
Font	Sans Serif 🗨							
Color Mode	Default							
Colour								
Text Rotation	0							
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 Vector $\stackrel{\Rightarrow}{\Rightarrow}$ in the ANSYS Workbench 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 Vector 1											
Geometry C	olor	or Symbol Render View									
Domains All Domains											
Definition											
Locations	symm	netry central	unit		▼						
Sampling	Equa	lly Spaced			•						
# of Points	1000	10000									
Variable	Veloc	ity			•						
Boundary Data		🔘 Hybrid		Ons	servative						
Projection	Tange	ential			•						

- i. Select All Domains from the Domains drop-down list.
- ii. Select symmetry central unit from the Locations drop-down list.
- iii. Select Equally Spaced from the Sampling drop-down list.
- iv. Set the **# of Points** to 10000.

Details of Vector 1

- v. Select Tangential from the Projection drop-down list.
- d. In the Details view for Vector 1, under the Color tab, configure the following settings.

Geometry	Color S	ymbol	Render	View		
Mode	Variable				•]
Variable	Pressure				•	
Range	Global				•	
Min				u	nknown	
Max				u	nknown	
Boundary Dat	a 🔘	Hybrid		Cons	servative	:
Color Scale	Linear				•	
Color Map	Default (F	Rainbow)		•	B
Undef. Color						

i. Select Variable from the Mode drop-down list.

- ii. Select Pressure from the Variable drop-down list.
- e. In the **Details** view for **Vector 1**, under the **Symbol** tab, configure the following settings.

Details of Vector 1									
Geometry	Color	Symbol	Render	View					
Symbol	Line A	Line Arrow							
Symbol Size	0.05								
V Normalize Symbols									

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

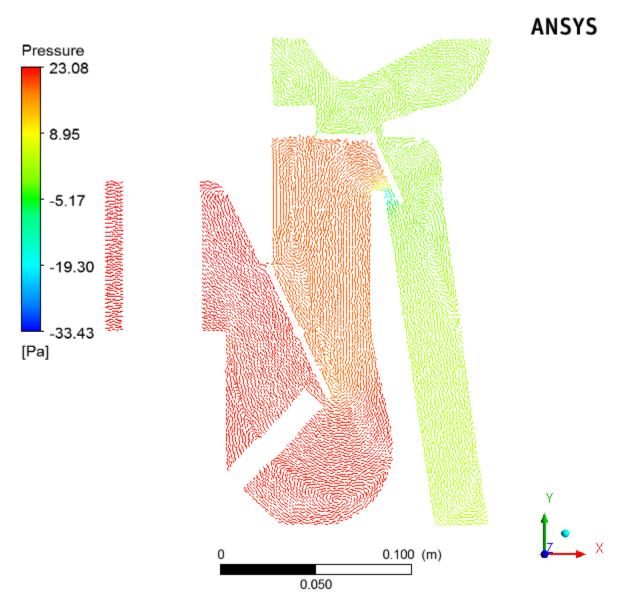
Vector 1 appears under User Locations and Plots in the Outline tree view.

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. 106).

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.

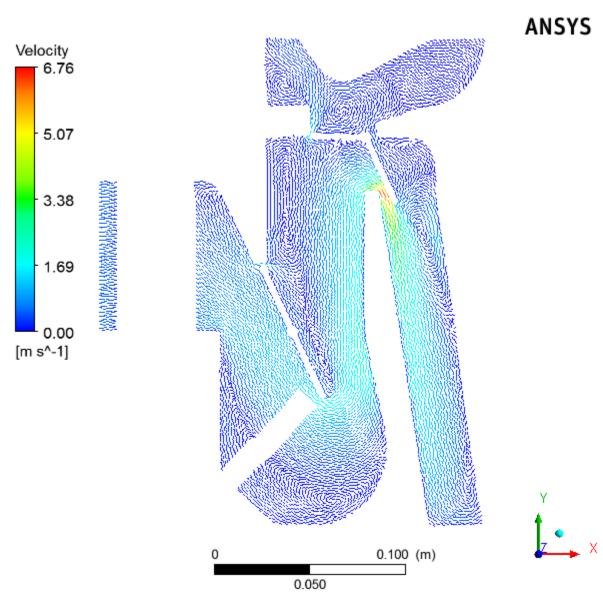
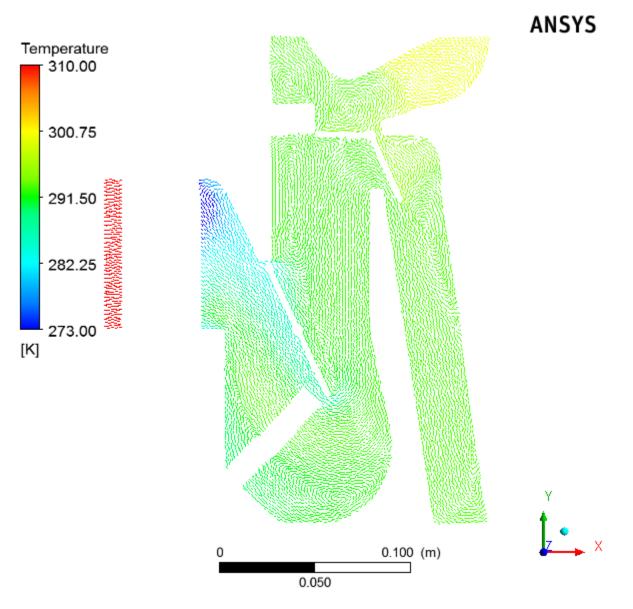


Figure 2.17: Vectors Colored by Velocity

- 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 three surface groups.

Surface groups are collections of surface locations in CFD-Post. In this tutorial, several surface groups are created in CFD-Post that will represent all of the outlets, all of the foot 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** tab open in the CFD-Post tree view, open the **Insert Surface Group** dialog box.

Insert → Location → Surface Group

🎯 Ins	ert Surface Group 📑 🞫
Name	alloutlets
	DK Cancel

- ii. Enter alloutlets for the Name of the surface group, and 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 ____ next to **Locations** to display the **Location Selector** dialog box.

💿 Location Selector							
Fluid Flow FLUENT							
Ĵ‡ inlet air							
1 interior fluid central unit fluid evaporator fluid central unit							
1 interior fluid central unit fluid evaporator fluid evaporator							
1 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							
]‡ outlet windshield							
Ĵ‡ symmetry central unit							
ĵ‡ symmetry evaporator							
ĵ‡ symmetry heat exchanger							
ĵ‡ wall central unit							
ĵ‡ wall evaporator							
ĵ‡ wall ftpos							
ĵ‡ wall hcpos							
ĵ‡ wall heat exchanger							
ĵ‡ wall inlet flow diverter							
ĵ‡ wall wsfpos							
<u>O</u> K <u>C</u> ancel							

- iv. In the Location Selector dialog box, select all of the outlet surfaces (outlet foot left, outlet front mid, outlet front side left, and outlet windshield) 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 close the Insert Expression dialog box.

Details of flou	tfront								
Definition	Plot	Evaluate							
-(massFlow	-(massFlow()@frontoutlets)*2								
Value		-0.000	264112 [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.

Perform the same steps as described above to create an expression called floutwindshield with the following definition:

-(massFlow()@outlet windshield)*2

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.

Perform the same steps as described above to create an expression called floutfoot with the following definition:

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

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.

Perform the same steps as described above to create an expression called outlettemp with the following definition:

massFlowAveAbs(Temperature)@alloutlets

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, 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. Figure 2.19: The Updated Project Loaded into ANSYS Workbench Displaying Parameter Outline, Properties, and Files View

A fluent-workbench-param - Workbench						
File View Tools Units Extensions He	elp					
🞦 📴 🛃 📑 Project						
Import 🖓 Reconnect 🔮 Refresh Project	t 🍠 Up	date Project 🀬 Resume 💔 Update All Des	ign Points			
Toolbox 💌 🗭 🗙	Project	Schematic				* 0 X
Analysis Systems						*
Fluid Flow - Blow Molding (Polyflow)						
Fluid Flow-Extrusion (Polyflow)		▼ A				
Fluid Flow (Fluent)		1 C Fluid Flow (FLUENT)				
Fluid Flow (Polyflow)		2 G Geometry				
🔀 IC Engine		3 🍘 Mesh 🗸				
C Throughflow						
E Component Systems		4 💽 Setup 🗸				1
🚓 BladeGen		5 🗑 Solution 🗸 🖌				
External Data		6 🥩 Results 🛛 🗸 🖌				
External Model	->	7 Parameters				
Finite Element Modeler	1.	Fluid Flow (FLUENT)				
Fluent		Hard How (FEDENT)				
Fluent (with TGrid meshing)						
🤪 Geometry						
🍘 Mesh		V				
Microsoft OfficeExcel	lip⊋ P	Parameter Set				-
Polyflow	Files					* 0 X
Polyflow - Blow Molding	r iitea				-	
Polyflow-Extrusion		A	B	с	D	E
Results	1	Name	Cel 🕞	Size 💌	Type 💌	Date Modified
System Coupling	_		ID 🛄			E
Vista AFD	2	A fluent-workbench-param.wbpj		332 KB	Workbench Project	7/25/2013 5:19:26 PM
Vista CCD						
Vista CCD (with CCM)	3	Geom.agdb	A2	2 MB	Geometry File	1/7/2013 3:14:52 PM
Vista CPD .		â				
Vista RTD	4		A2	2 MB	Geometry File	7/25/2013 2:35:56 PM
Vista IP Design Exploration						
	5	FFF.mshdb	A3	23 MB	.mshdb	7/25/2013 2:39:00 PM
External Connection Systems						
View All / Customize		FFF water and				Tincing a second set
	•					· ·
Ready					Show Progress	Show 4 Messages

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 menu.

2.4.6. 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)

	A	8	с	D	Ε	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	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 **Exported** column in the table of design points, and select the check box in the row for the duplicated design point **DP 1** (cell N4).

This allows the data from this new design point to be saved to a separate project for analysis.

- c. Click **OK** to acknowledge the information message prompting you to update the design points in order for the design points to be exported.
- 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 the ability to export its data as well.

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

	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	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	of Design Poi	ts (with DP0)	, DP1, and I	DP2 Defined)
-----------------------	---------------	---------------	--------------	--------------

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

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.

When the design points have been updated, note the addition of two more ANSYS Workbench project files (and their corresponding folders) in your current working directory (fluent-workbench-param_dpl.wbpj and fluent-workbench-param_dp2.wbpj). You can open each of these projects up separately and examine the results of each parameterized simulation. If you make any changes to the design point and update the design point, then an additional ANSYS Workbench project is created (for example, fluent-workbench-param_dp1_1.wbpj).

6. 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)

	A	E		G	н	1	3	к	L	м
1	Name 💌	P4 - input_hcpos 💌	P5 - input_ftpos 💌	P6 - input_valipos 💌	P7 - in_velocity 💌	P8 - in_temp 💌	P9 - floutfront 💌	P10 - foutwindshield 💌	P11 - floutfoot 💌	P12 - outlettemp 💌
2	Units				n s^-1 💌	к 💌	kg s^-1	kg s^-1	kg s^-1	K
3	Current	15	25	90	0.5	310	-0.00026264	0.0011266	0.010161	292.34
4	DP 1	45	45	45	0.6	300	9.9634E-05	0.0071286	0.0060017	284.98
5	DP 2	90	60	15	0.7	290	0.0016706	0.0081472	0.0056171	277.13

- 7. Click the **Project** tab, just above the ANSYS Workbench toolbar to return to the **Project Schematic**.
- 8. 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 enabled the **Exported** option when setting them up.

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

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

10. Quit ANSYS Workbench.

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

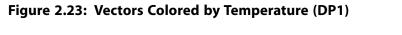
2.4.7. Postprocessing the New Design Points in CFD-Post

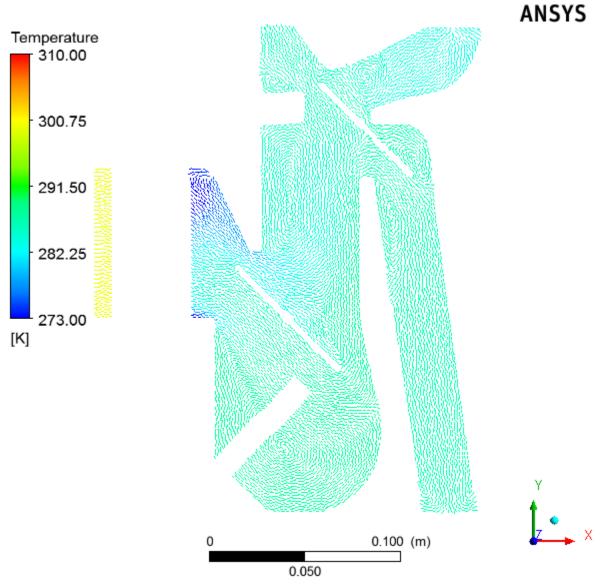
In this step, you will open the ANSYS Workbench project for each of the design points and inspect the vector plots based on the new results of the simulations.

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

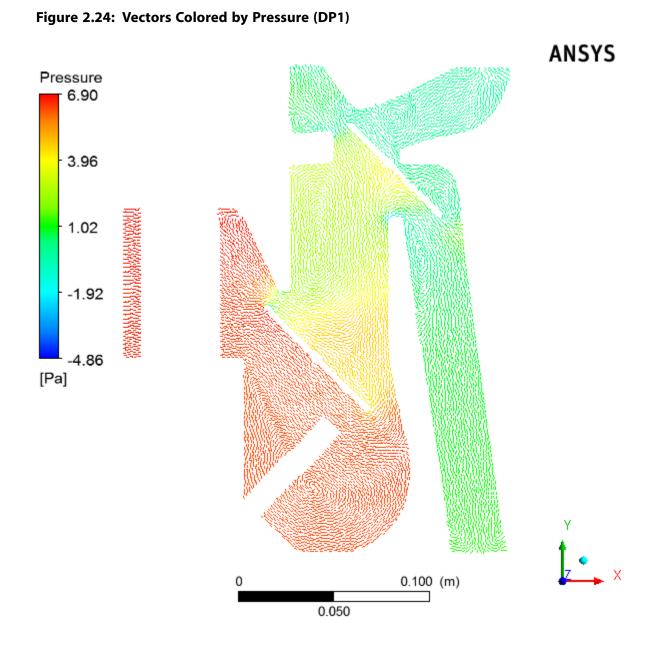
In your current working folder, double-click the fluent-workbench-param_dpl.wbpj file to open ANSYS Workbench.

- 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.



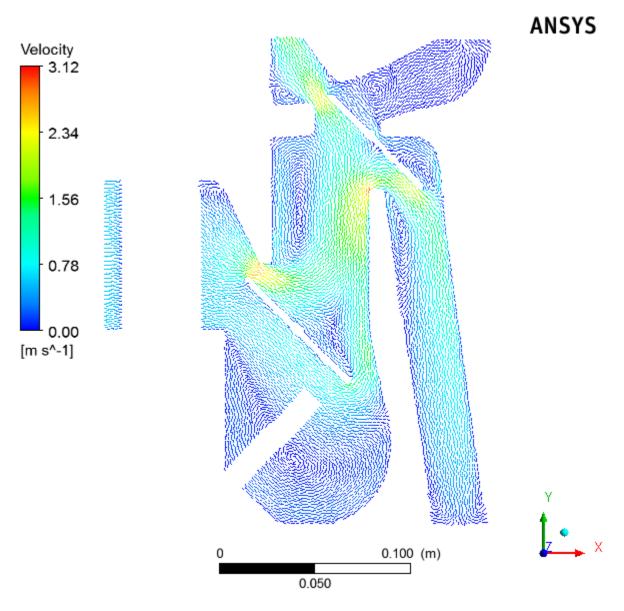


d. View the vector plot colored by pressure. Ensure that **Range** in the **Color** tab is set to **Global**.



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.

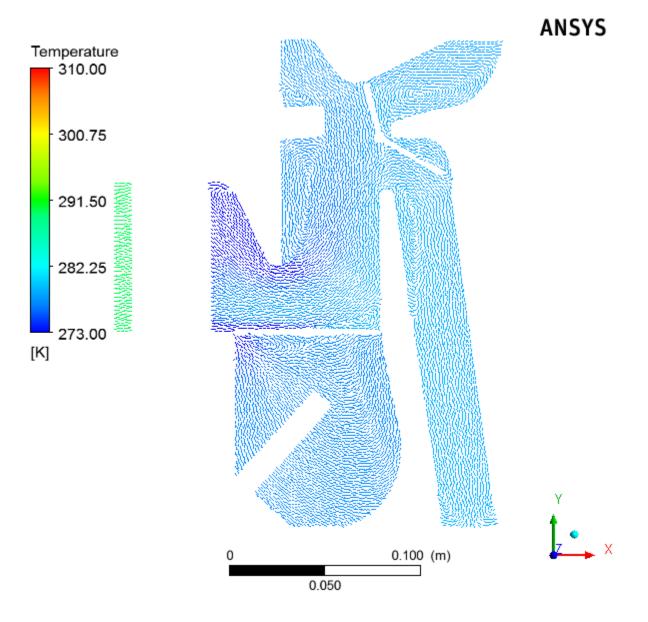


Figure 2.26: Vectors Colored by Temperature (DP2)

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

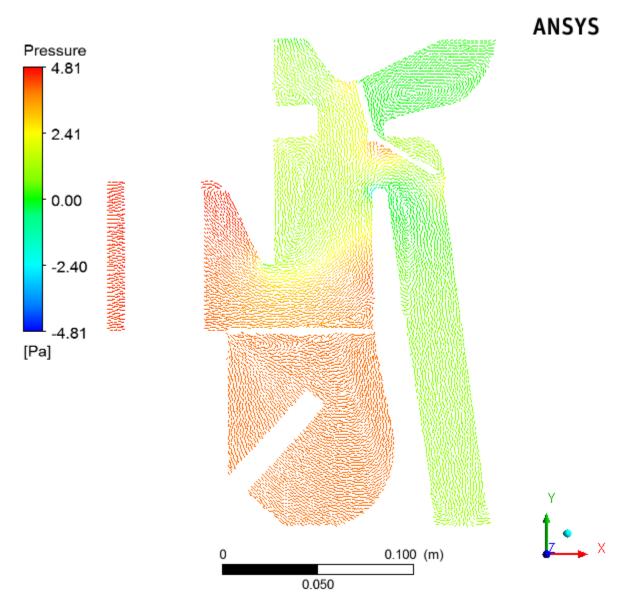


Figure 2.27: Vectors Colored by Pressure (DP2)

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

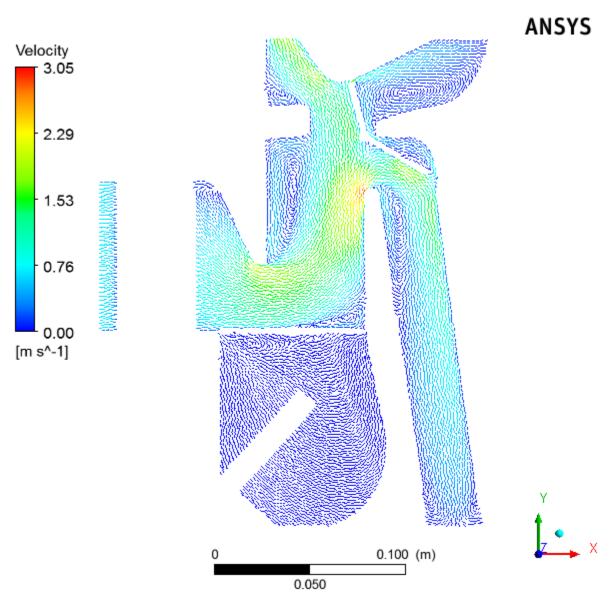


Figure 2.28: Vectors Colored by Velocity (DP2)

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.

2.4.8. 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
- 3.5. 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 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.
- Set up a surface monitor 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.

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

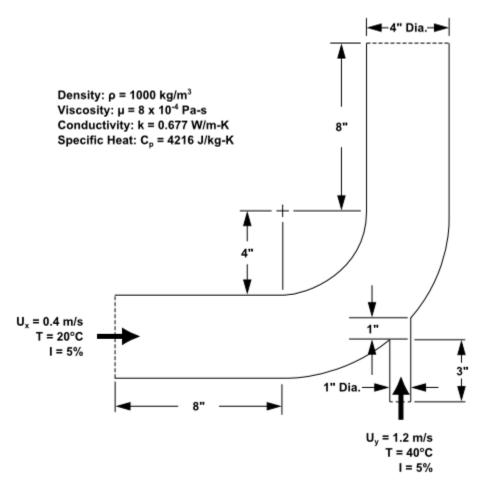
The problem to be considered is shown schematically in Figure 3.1: Problem Specification (p. 124). A cold fluid at 20° C flows into the pipe through a large inlet, and mixes with a warmer fluid at 40° C that

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.

Figure 3.1: Problem Specification



3.4. Setup and Solution

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

- 3.4.1. Preparation
- 3.4.2. Launching ANSYS Fluent
- 3.4.3. Reading the Mesh
- 3.4.4. General Settings
- 3.4.5. Models
- 3.4.6. Materials
- 3.4.7. Cell Zone Conditions
- 3.4.8. Boundary Conditions
- 3.4.9. Solution
- 3.4.10. Displaying the Preliminary Solution

3.4.11. Using the Coupled Solver 3.4.12. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip the introduction_R150.zip file you downloaded to your working directory. This file contains a folder, introduction, that holds the file elbow.msh that you will use in this tutorial. The introduction directory also contains a solution_files sub-folder that contains the solution files created during the preparation of 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 or graphics card.

3.4.2. Launching ANSYS Fluent

1. Open the Fluent Launcher by clicking the Windows **Start** menu, then selecting **Fluent 15.0** in the **Fluid Dynamics** sub-menu of the **ANSYS 15.0** program group.

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

Start \rightarrow All Programs \rightarrow ANSYS 15.0 \rightarrow Fluid Dynamics \rightarrow Fluent 15.0

Eluent Launcher	
ANSYS	Fluent Launcher
Dimension ② 2D ③ 3D	Options Double Precision Meshing Mode
Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme	Processing Options Serial Parallel
🛨 Show More Options	
<u> </u>	<u>C</u> ancel <u>H</u> elp ▼

- 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. Enable the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options.

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.

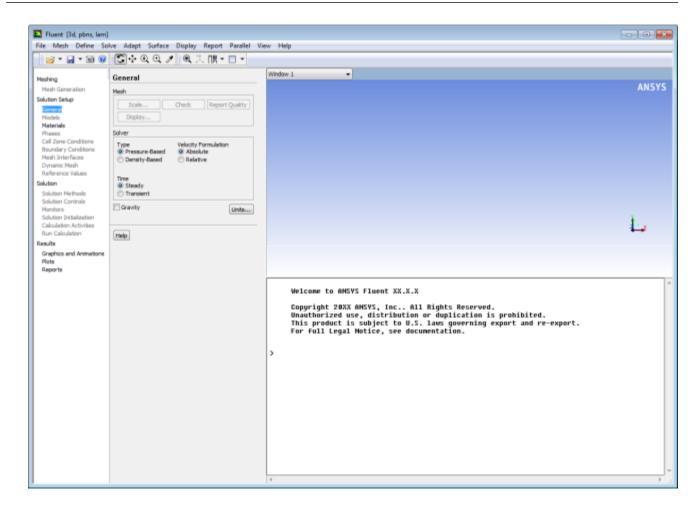
3. Set the working path to the directory created when you unzipped introduction_R150.zip.

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

Alternatively, you can click the browse button (^[26]) next to the **Working Directory** text box and browse to the directory, using the **Browse For Folder** dialog box.

Iluent Launcher				
ANSYS	Fluent Launcher			
Dimension 2D 3D Display Options Display Mesh After Reading Comparison Windows Workbench Color Scheme	Options Double Precision Processing Options Serial Parallel			
General Options Parallel Settings	Scheduler Environment			
Version Working Directory C:\Tutorials\introduction Fluent Root Path C:\PROGRA~1\ANSYSI~1\xXX\fluent Image: Use Journal File				
<u>K</u> efa	ault <u>C</u> ancel <u>H</u> elp –			

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



3.4.3. Reading the Mesh

1. Read the mesh file elbow.msh.

 $\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

Select Read from the File menu, then select Mesh... to open the Select File dialog box.

Select File					x
Look in:	introduction	-	G 🦻	⊳ 🔝	
Recent Places	elbow.msh				
Desktop					
Libraries					
Computer					
Network	Mesh File	elbow.msh		•	ОК
	Files of type:	Mesh Files		•	Cancel
	📝 Display Mesh Af	ter Reading			

- a. Select the mesh file by clicking elbow.msh in the introduction directory 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 (\nearrow) 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:

Table 3.1: View Manipulation Instructions

Action	Using Graphics Toolbar Buttons and the Mouse	
Rotate view (vertical, hori- zontal)	After clicking the Rotate icon, S , 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 hori- zontal axis.	
Translate or pan view	After clicking the Pan 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 Zoom in/Out icon, (Q , press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.	
Zoom to se- lected area	After clicking the Zoom to Area icon, \textcircled{P} , 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.	

Clicking the **Fit to Window** icon, ((Rec), 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 **S**.

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. 132).

Select Graphics and Animations in the navigation pane, then click Views... in the Graphics and Animations task page.

Graphics and Animations → Views...

💶 Views		— ×		
Views back bottom front isometric left right top	Actions Default Auto Scale Previous Save Delete Read	Mirror Planes 🖹 🗐 🗐		
Save Name front	Write	Define		
Apply Camera Close Help				

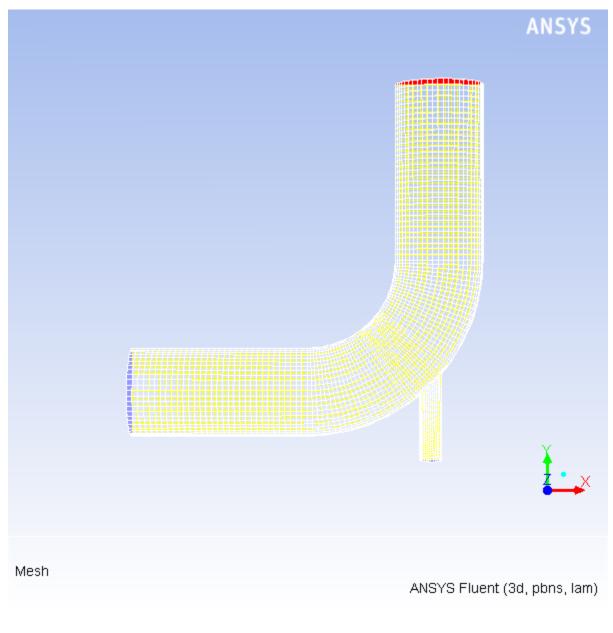
a. Select **front** from the **Views** selection list.

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 zight as follows:

- Left-click an axis to point it in the positive direction.
- Right-click an axis to point it in the negative direction.
- Left-click the iso-ball to set the isometric view.

3.4.4. General Settings

Select **General** in the navigation pane to perform the mesh-related activities and to choose a solver.

Meshing	General
Mesh Generation	Mesh
Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Scale Check Report Quality Display Display Solver Type Velocity Formulation Image: Pressure-Based Image: Absolute Image: Density-Based Image: Relative
Solution	Time Steady
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities	Transient Gravity Units
Run Calculation	Help
Results	
Graphics and Animations Plots Reports	

1. Check the mesh.

$\mathbf{O}_{\mathsf{General}} \rightarrow \mathsf{Check}$

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

```
Domain Extents:
    x-coordinate: min (m) = -8.000000e+000, max (m) = 8.000000e+000
    y-coordinate: min (m) = -9.134633e+000, max (m) = 8.000000e+000
    z-coordinate: min (m) = 0.000000e+000, max (m) = 2.000000e+000
Volume statistics:
    minimum volume (m3): 5.098270e-004
    maximum volume (m3): 2.330737e-002
        total volume (m3): 1.607154e+002
Face area statistics:
    minimum face area (m2): 4.865882e-003
    maximum face area (m2): 1.017924e-001
Checking mesh......
Done.
```

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, 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.

$\mathbf{\mathbf{\dot{\mathbf{\nabla}}}} \mathbf{General} \rightarrow \mathbf{Scale...}$

Scale Mesh		—	
Domain Extents		Scaling	
Xmin (in) -8	Xmax (in) 8	 Convert Units Specify Scaling Factors 	
Ymin (in) -9.134633	Ymax (in) 8	Mesh Was Created In	
Zmin (in)	Zmax (in) 2	Scaling Factors	
View Length Unit In		× 0.0254	
[in 🗸		Y 0.0254	
		Z 0.0254	
		Scale Unscale	
Close Help			

- a. Ensure that **Convert Units** is selected in the **Scaling** group box.
- b. Select **in** from the **Mesh Was Created In** drop-down list 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 dialog box above.
- 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 **General** task page and make the appropriate change, in the **Set Units** dialog box.

3. Check the mesh.



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. Retain the default settings of pressure-based steady-state solver in the **Solver** group box of the **General** task page.

3.4.5. Models

Models
Models
Multiphase - Off Energy - Off Viscous - Laminar Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Solidification & Melting - Off Acoustics - Off Eulerian Wall Film - Off
Edit
Help

1. Enable heat transfer by activating the energy equation.

Models →	Ē	 Energy →	Edit
----------	---	--------------	------

💶 Energy 💽
Energy Energy Equation
OK Cancel Help

- a. Enable the **Energy Equation** option.
- b. Click **OK** to close the **Energy** dialog box.
- 2. Enable the k- ε turbulence model.

 $\texttt{OMODELS} \rightarrow \texttt{Edit...}$

Viscous Model	×
Model Inviscid Laminar Spalart-Allmaras (1 eqn) K-epsilon (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (7 eqn) Scale-Adaptive Simulation (DES) Detached Eddy Simulation (DES) Large Eddy Simulation (LES) c-epsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Scalable Wall Functions Enhanced Wall Treatment User-Defined Wall Functions Enhanced Wall Treatment Options Pressure Gradient Effects Thermal Effects	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number None Energy Prandtl Number None Current Compone Co

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

The Viscous Model dialog box will expand.

- b. Retain the default selection of **Standard** in the **k-epsilon Model** group box.
- c. 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 Setting Up the k- ε Model in the User's Guide.

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

3.4.6. Materials

Materials
Materials
Fluid air Solid aluminum
Create/Edit Delete
Help

1. Create a new material called **water**.



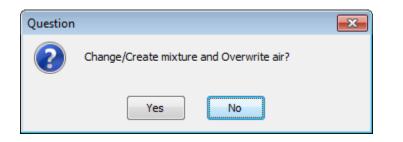
Create/Edit Materials					×
Name water		aterial Type luid		•	Order Materials by
Chemical Formula	FL	uent Fluid Materi	als		Chemical Formula
1	w	vater		-	Fluent Database
	M	ixture			User-Defined Database
	n	one		Ŧ	
Properties					
Density (kg/m3)	constant		▼ Edit	Â	
	1000				
Cp (Specific Heat) (j/kg-k)	constant		▼ Edit		
	4216			E	
Thermal Conductivity (w/m-k)	constant		▼ Edit		
	0.677				
Viscosity (kg/m-s)	constant		▼ Edit		
	0.0008				
,			(
	Change/Create	Delete	Close	Help	

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

Property	Value
Density	1000 kg/m^3
c _p	4216 <i>J/kg-K</i>
Thermal Conductivity	0.677 <i>W/m-K</i>
Viscosity	8e-04 <i>kg/m-s</i>

c. 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 list of materials that originally contained only **air**.



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.

d. 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.

e. Close the Create/Edit Materials dialog box.

3.4.7. Cell Zone Conditions

.

- ----

Cell Zone Con	ditions
Zone	
fluid	
1	
Phase	Type ID
mixture	→ fluid → 2
Edit	Copy Profiles
Parameters	Operating Conditions
Display Mesh	
Porous Formulation	1
Superficial Velo	
Physical Velocit	У
Help	

1. Set the cell zone conditions for the fluid zone (**fluid**).



I Fluid
Zone Name
fluid
Material Name water
Frame Motion Laminar Zone Source Terms
Mesh Motion Fixed Values
Porous Zone
Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Multiphase
Rotation-Axis Origin Rotation-Axis Direction
X (in) 0 constant V 0 constant V
Y (in) 0 constant V 0 constant V
Z (in) 0 constant V Z 1 constant V
OK Cancel Help

- a. Select water from the Material Name drop-down list.
- b. Click **OK** to close the **Fluid** dialog box.

3.4.8. Boundary Conditions

Boundary Con	ditions	
Zone		
default-interior pressure-outlet-7 symmetry		
velocity-inlet-5		
velocity-inlet-6 wall		
Phase	Type	ID 7
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh	Periodic Conditions	
Highlight Zone		
Help		

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

$\textcircled{Boundary Conditions} \rightarrow \fbox{velocity-inlet-5} \rightarrow \texttt{Edit...}$

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 (²²) 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.

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

Velocity Inlet				
Zone Name				
velocity-inlet-5				
Momentum Thermal Radiation Species	DPM Multiphase U	DS		
Velocity Specification Method	Components	•		
Reference Frame	Absolute	•		
Supersonic/Initial Gauge Pressure (pascal)	0	constant 💌		
Coordinate System	Cartesian (X, Y, Z)			
X-Velocity (m/s)	0.4	constant 🔻		
Y-Velocity (m/s)	0	constant 🔹		
Z-Velocity (m/s)	0	constant 🔻		
Turbulence				
Specification Method	ntensity and Hydraulic Diam	eter 🔻		
	Turbulent Intensity (%	6) 5 P		
	Hydraulic Diameter (i	1) 4		
		•		
OK Cancel Help				

a. Select Components from the Velocity Specification Method drop-down list.

The Velocity Inlet dialog box will expand.

- b. Enter 0.4 m/s for **X-Velocity**.
- c. Retain the default value of 0 m/s for both **Y-Velocity** and **Z-Velocity**.
- d. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
- e. Retain the default value of 5 % for **Turbulent Intensity**.
- f. 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.

g. Click the **Thermal** tab.

🛂 Velocity Inlet 🧮	
Zone Name	
velocity-inlet-5	
Momentum Thermal Radiation Species DPM Multiphase UDS	1
Temperature (k) 293.15 constant	
OK Cancel Help	

- h. Enter 293.15 K for **Temperature**.
- i. Click **OK** to close the **Velocity Inlet** dialog box.
- 2. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-6**), using the values in the following table:

Boundary Conditions →	\mathbf{F} velocity-inlet-6 \rightarrow Edit
-----------------------	--

Velocity Specification Method	Components
X-Velocity	0 <i>m/s</i>
Y-Velocity	1.2 <i>m/s</i>
Z-Velocity	0 <i>m/s</i>
Specification Method	Intensity and Hydraulic Diameter
Turbulent Intensity	5%
Hydraulic Diameter	1 inch
Temperature	313.15 <i>K</i>

3. Set the boundary conditions at the outlet (**pressure-outlet-7**), as shown in the **Pressure Outlet** dialog box.

Ô	a 11.1		
Boundary	Conditions \rightarrow	🔄 pressure-out	liet-/ → Edit

Pressure Outlet
Zone Name
pressure-outlet-7
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method Normal to Boundary
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 (in) 4
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.

4. For the wall of the pipe (wall), retain the default value of $0 W/m^2$ for **Heat Flux** in the **Thermal** tab.



💶 Wall			×
Zone Name			· · · · · · · · · · · · · · · · · · ·
wall		1	
Adjacent Cell Zone			
fluid			
Momentum Thermal Rad	diation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
Heat Flux	Heat Flux (w/m2	2) 0	constant 🔹
 Temperature Convection 		Wall Thickness	(in) 0
Radiation	Heat Generation Rate (w/m3	2)	•••••••••••••••••••••••••••••••••••••••
 Mixed via System Coupling 	Heat Generation Rate (w/m.	0	constant 👻
Material Name			Shell Conduction
aluminum	✓ Edit		
	OK	Help	

3.4.9. Solution

In the steps that follow, you will set up and run the calculation using the task pages listed under the **Solution** heading in the navigation pane.

1. Select a solver scheme.



Leave the **Scheme** at the default **SIMPLE** for this calculation.

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 👻	Ŧ
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
High Order Term Relaxation Options	
Default	
Help	

2. Enable the plotting of residuals during the calculation.

 $\textcircled{} Monitors \rightarrow \overleftarrow{\sqsubseteq} Residuals \rightarrow Edit...$

Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria	<u> </u>
V Plot	continuity	V		0.001	
Window	x-velocity			0.001	E
Iterations to Plot	y-velocity			0.001	
1000	z-velocity			0.001	-
	Residual Values			Convergence C	riterion
Iterations to Store	Normalize		Iterations	absolute	•
	Scale				
	Compute Loca	al Scale			
OK Plot Renormalize Cancel Help					

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Leave the **Absolute Criteria** of **continuity** at the default level of 0.001 as shown in the **Residual Monitors** dialog box.
- c. 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. It is a good practice to also define a surface monitor that can help evaluate whether the solution is truly converged. You will do this in the next step.

3. Define a surface monitor of average temperature at the outlet (pressure-outlet-7).

♦ Monitors → Create... (Surface Monitors)

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

Surface Monitor				
Name outlet-temp-avg	Report Type Mass-Weighted Average			
Options	Field Variable Temperature			
Vindow	Static Temperature			
2	default-interior pressure-outlet-7 symmetry			
File Name Outlet-temp-avg.out X Axis	velocity-inlet-5 velocity-inlet-6 wall			
Get Data Every				
3 Iteration				
	☐ Highlight Surfaces New Surface ▼			
OK Cancel Help				

- a. Enter outlet-temp-avg for the Name of the surface monitor.
- b. Enable the Plot and Write options for outlet-temp-avg.
- c. Enter outlet-temp-avg.out for File Name.
- d. Set Get Data Every to 3 by clicking the up-arrow button.

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

- e. Select Mass-Weighted Average from the Report Type drop-down list.
- f. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- g. Select **pressure-outlet-7** from the **Surfaces** selection list.
- h. Click OK to save the surface monitor settings and close the Surface Monitor dialog box.

The name and report type of the surface monitor you created will be displayed in the **Surface Monitors** selection list in the **Monitors** task page.

4. Set a convergence monitor for **outlet-temp-avg**.

Monitors → Convergence Manager...

Convergence Manager			.	-
Active Monitor Name Image: Construction of the second se	Stop Criterion 1e-5	Initial Iterations to Ignore Previous Iterations to Consider Average Over 20 15 1 1	Print 🖉	
Choose Condition Eve All Conditions are Met Any Condition is Met	ry Iteration	OK Cancel Heb		

- a. Activate the convergence criterion on outlet temperature by checking the **Active** box next to **outlet-temp-avg**.
- a. Enter 1e-5 for **Stop Criterion**.
- b. Enter 20 for Initial Iterations to Ignore.
- c. Enter 15 for **Previous Iterations to Consider**.
- d. Enable Print.
- e. Enter 3 for Every Iteration
- f. Click **OK** to save the convergence monitor settings and close the **Convergence Manager** dialog box.

These settings will cause Fluent to consider the solution converged when the monitor value for each of the previous 15 iterations is within 0.001% of the current value. Convergence of the monitor 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.

5. Initialize the flow field.

Over Solution Initialization

Solution Initialization					
Initialization Methods Hybrid Initialization Standard Initialization 					
More Settings Initialize					
Reset DPM Sources Reset Statistics					
Help					

a. Leave the Initialization Method at the default Hybrid Initialization.

b. Click Initialize.

6. Save the case file (elbow1.cas.gz).

Select File							×
Look in:	introduction		•	G	ø 🖻		
Recent Places							
Desktop							
Libraries							
Computer							
Network	Case File	elbow1.cas.gz			•	10	
	Files of type:	Case Files			•	Can	el
	📝 Write Binary File	s					

a. (optional) Indicate the directory in which you would like the file to be saved.

By default, the file will be saved in the directory from which you read in elbow.msh (that is, the introduction directory). You can indicate a different directory by browsing to it or by creating a new directory.

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.
- 7. Start the calculation by requesting 150 iterations.

Run Calculation

Run Calculation
Check Case Preview Mesh Motion
Number of Iterations Reporting Interval
Profile Update Interval
Data File Quantities Acoustic Signals
Calculate
Help

- a. Enter 150 for Number of Iterations.
- b. Click Calculate.

Note

By starting the calculation, you are also starting to save the surface monitor data at the rate specified in the **Surface monitors** dialog box. If a file already exists in your working directory with the name you specified in the **Define Surface Monitor** 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 monitor history will be plotted in graphics window 2 (Figure 3.3: Convergence History of the Mass-Weighted Average Temperature (p. 154)).

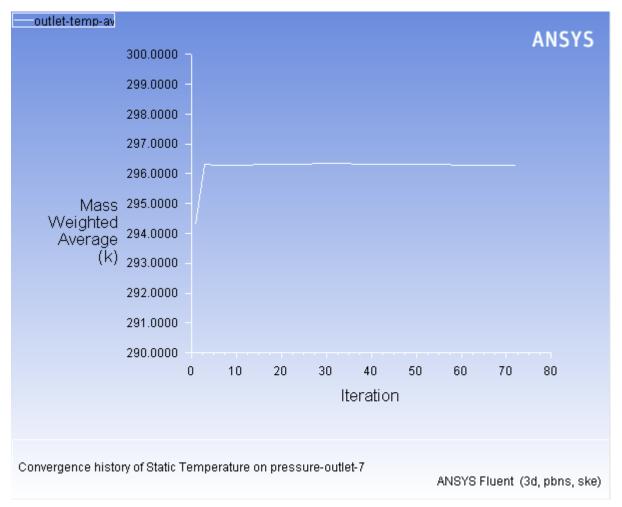
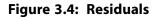
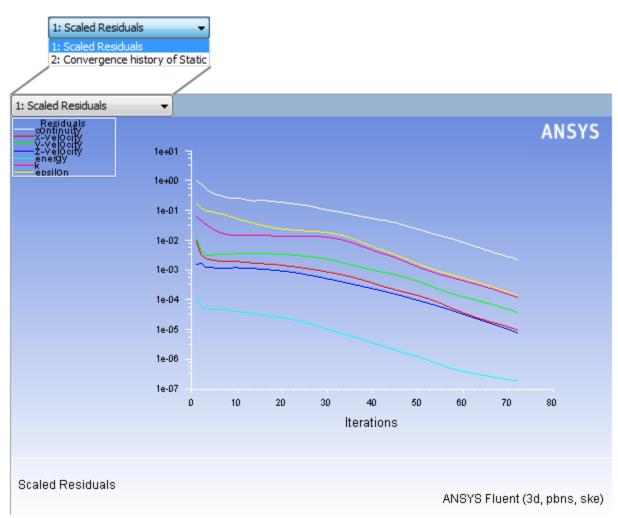


Figure 3.3: Convergence History of the Mass-Weighted Average Temperature

Similarly, the residuals history will be plotted in window 1 in the background. You can display the residuals history by selecting window 1 from the graphics window drop-down list (Figure 3.4: Residuals (p. 155)).





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 monitor converges to within the tolerance specified in the **Convergence Manager** 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 75 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.

8. Examine the plots for convergence (Figure 3.3: Convergence History of the Mass-Weighted Average Temperature (p. 154) and Figure 3.4: Residuals (p. 155)).

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.

9. Examine the mass flux report for convergence.

 $\diamondsuit \mathsf{Reports} \to \blacksquare \mathsf{Fluxes} \to \mathsf{Set} \mathsf{Up...}$

¢				
E Flux Reports		×		
Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types axis exhaust-fan fan inlet-vent Boundary Name Pattern Save Output Parameter	Boundaries default-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall	Results -1.9193115234375 1.617520928382874 0.3018066585063934 Image: mail of the second se		
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.

10. 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.

3.4.10. Displaying the Preliminary Solution

In the steps that follow, you will visualize various aspects of the flow for the preliminary solution, using the task pages listed under the **Results** heading in the navigation pane.

1. Display filled contours of velocity magnitude on the symmetry plane (Figure 3.5: Predicted Velocity Distribution after the Initial Calculation (p. 159)).



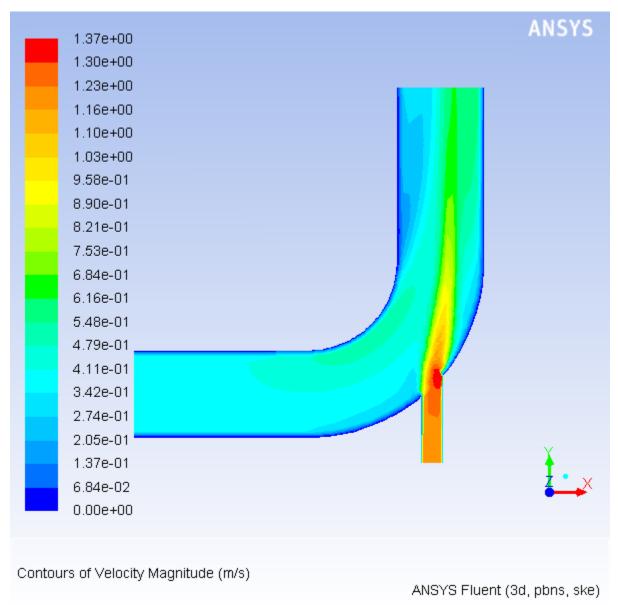
Contours	••••			
Options	Contours of			
▼ Filled	Velocity			
V Node Values				
Global Range	Velocity Magnitude 👻			
V Auto Range	Min Max			
Clip to Range	0 0			
Draw Profiles				
Draw Mesh	Surfaces 🗵 🗏 🗏			
	default-interior			
Levels Setup	pressure-outlet-7			
	symmetry 🗧			
20 🔺 1 🗭				
	velocity-inlet-6			
Surface Name Pattern	New Surface 🔻			
Match				
	Surface Types			
	axis 🔺			
	dip-surf			
	exhaust-fan			
	fan 👻			
Display Compute 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,

 ${}^{\textcircled{M}}$, will cause the object to fit exactly and be centered in the window.

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. Display filled contours of temperature on the symmetry plane (Figure 3.6: Predicted Temperature Distribution after the Initial Calculation (p. 161)).

 $\label{eq:Graphics} Graphics and Animations \rightarrow \ensuremath{\overline{=}}^{\bullet} Contours \rightarrow Set Up...$

Contours				
Options	Contours of			
✓ Filled	Temperature			
Vode Values	Static Temperature			
 Global Range Auto Range 	Min Max			
Clip to Range				
Draw Profiles				
Draw Mesh	Surfaces 🗵 🗏 🚍			
	default-interior			
Levels Setup	pressure-outlet-7			
20 🔺 1 🔺	symmetry = velocity-inlet-5			
	velocity-inlet-6			
Surface Name Pattern				
Match	New Surface 💌			
Match	Surface Types			
	axis			
	dip-surf and a state of the sta			
	fan +			
	[<u> </u>			
Display Compute Close Help				

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

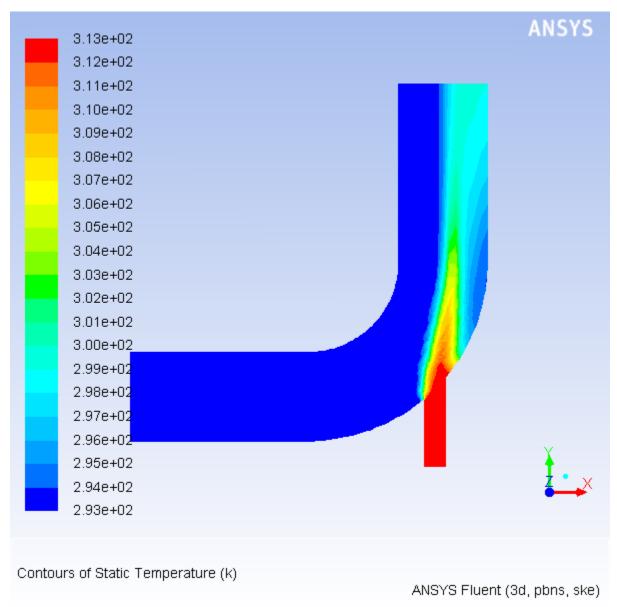


Figure 3.6: Predicted Temperature Distribution after the Initial Calculation

3. Display velocity vectors on the symmetry plane (Figure 3.9: Magnified View of Resized Velocity Vectors (p. 165)).

Graphics and Animations → \blacksquare Vectors → Set Up...

C Vectors			x	
Options	Vectors of			
Global Range	Velocity 🗸			
V Auto Range	Color by			
Clip to Range	Velocity			
Draw Mesh	Velocity Magnitude			
Style	Min (m/s)	Max (m/s)		
arrow 🔻	0.1385844	1.41918		
Scale Skip	Surfaces			
4 2	default-interior			
Vector Options	pressure-outlet-7			
	symmetry velocity-inlet-5			
Custom Vectors	velocity-inlet-6			
	wall			
Surface Name Pattern				
Match	New Surface 🔻			
	Surface Types			
	axis		*	
	clip-surf			
	exhaust-fan			
	fan -		Ψ.	
Display Compute Close Help				

- a. Select symmetry from the Surfaces selection list.
- b. Click **Display** to plot the velocity vectors.

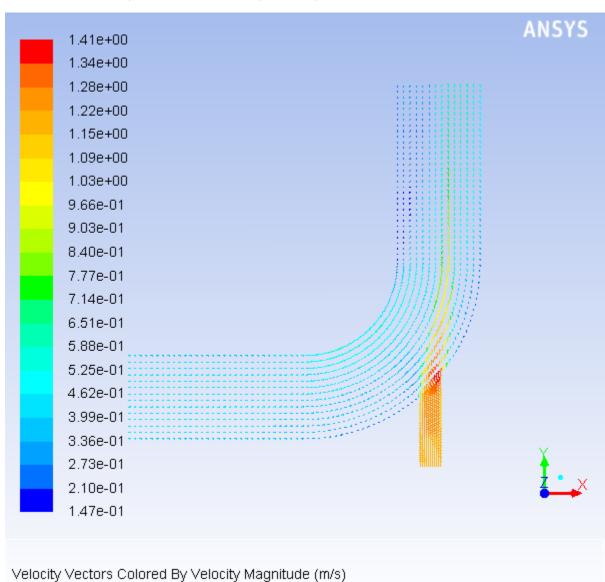


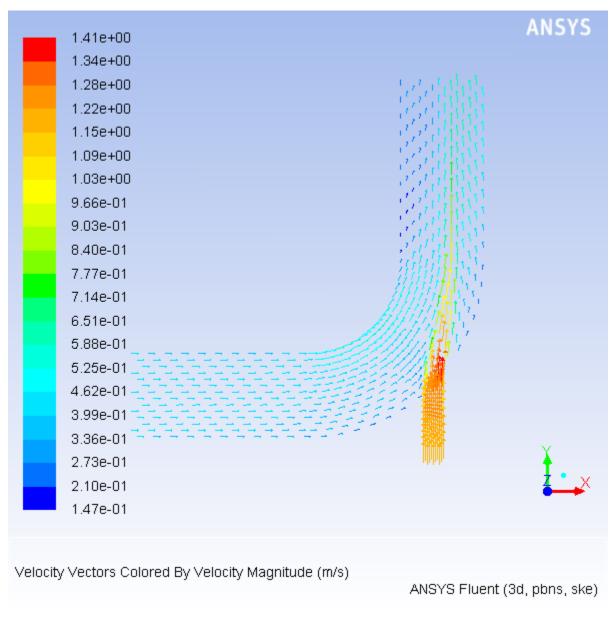
Figure 3.7: Velocity Vectors Colored by Velocity Magnitude

ANSYS Fluent (3d, pbns, ske)

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.

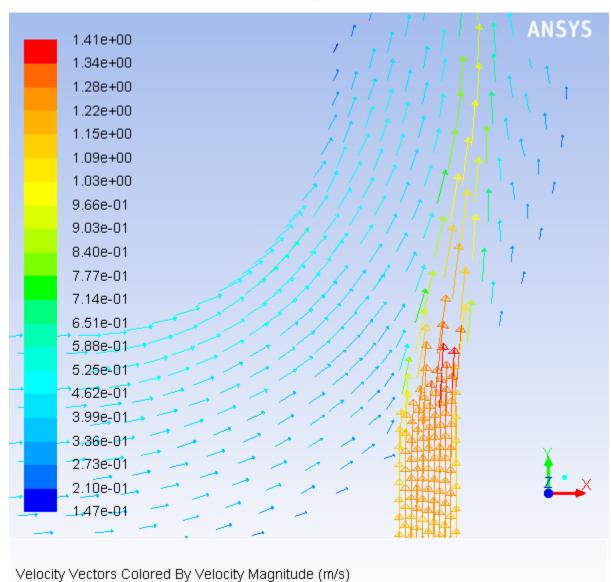
- c. Enter 4 for Scale.
- d. Set Skip to 2.
- e. Click **Display** again to redisplay the vectors.





- 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. 130). The image will be redisplayed at a higher magnification (Figure 3.9: Magnified View of Resized Velocity Vectors (p. 165)).





h. Zoom out to the original view.

You also have the option of selecting the original view in the **Views** dialog box:

$\clubsuit Graphics and Animations \rightarrow Views...$

Select front from the Views selection list and click Apply, then close the Views dialog box.

ANSYS Fluent (3d, pbns, ske)

Q Views		×
Views back bottom front isometric left right top Save Name front	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes = symmetry Define Plane Periodic Repeats Define
Apply Came	ra Close	Help

4. Create a line at the centerline of the outlet. For this task, you will use the **Surface** command that is at the top of the ANSYS Fluent window.

Iso-Surface		×
	 From Surface default-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall 	
Image: Second	From Zones	
Create Compute Manag	ge Close Help	

Surface → Iso-Surface...

- 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** boxes.

- 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** is added to the **From Surface** list in the dialog box.

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.

- 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)).

Solution XY Plot		
Options Image: Option on Values Image: Position on Values Image: Position on Values Image: Order Points	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Temperature Static Temperature X Axis Function Direction Vector Surfaces Use a state of the state of th
Plot	Axes	Curves Close Help

 $\textcircled{Plots} \rightarrow \blacksquare XY Plot \rightarrow Set Up...$

- 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... to open the Select File dialog box.
 - i. 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.

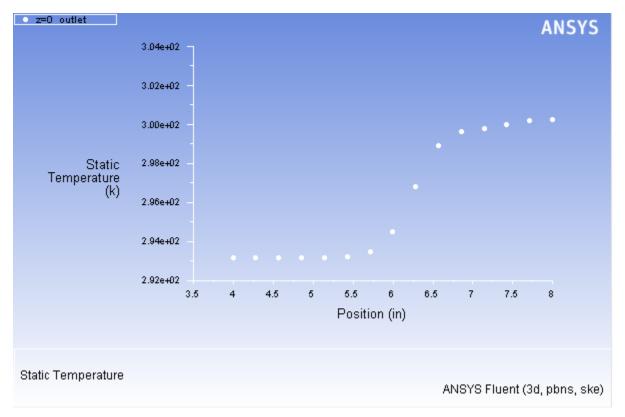


Figure 3.10: Outlet Temperature Profile for the Initial Solution

6. Define a custom field function for the dynamic head formula $(\rho \cdot |V|^2/2)$. For this task, you will use the **Define** menu that is at the top of the ANSYS Fluent window.



- 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.
- 7. Display filled contours of the custom field function (Figure 3.11: Contours of the Dynamic Head Custom Field Function (p. 171)).

Contours	×
Options	Contours of
V Filled	Custom Field Functions
✓ Node Values ✓ Global Range	dynamic-head 🗸
Auto Range	Min Max
Clip to Range	0 0
Draw Mesh	Surfaces
	default-interior
Levels Setup	pressure-outlet-7
	symmetry velocity-inlet-5
	velocity inlet 5
Surface Name Pattern	
Mat	New Surface
	Surface Types
	axis
	exhaust-fan
	fan 👻
Display	Compute Close Help

Graphics and Animations → \blacksquare Contours → Set Up...

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.

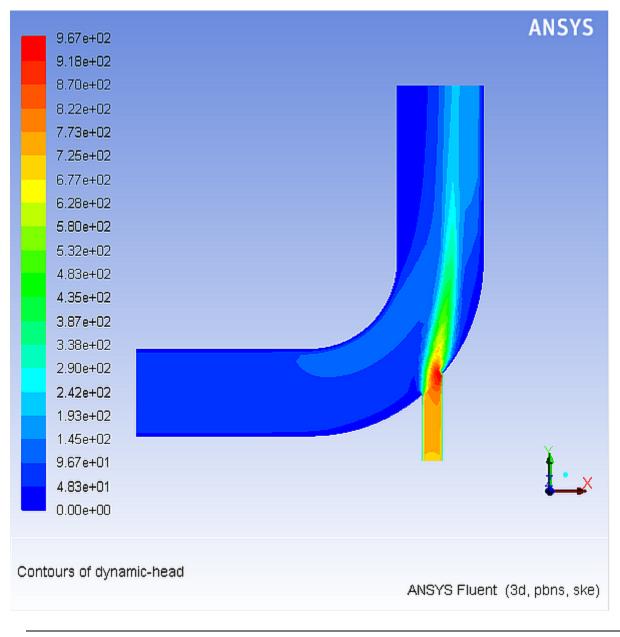


Figure 3.11: Contours of the Dynamic Head Custom Field Function

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 made earlier.

3.4.11. 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.

1. Change the solver settings.

Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled 🗸	
Spatial Discretization	
Gradient	Â.
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	
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.

Solution Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
More Settings Initialize
Patch
Reset DPM Sources Reset Statistics
Help

- a. Leave the Initialization Method at the default Hybrid Initialization.
- b. Click Initialize.
- 3. Run the solution for an additional 90 iterations.

Run Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Number of Iterations 90 💌	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- a. Ensure that 90 is entered for **Number of Iterations**.
- b. Click Calculate.

A dialog box will appear asking if you want to append data to outlet-temp-avg.out. Click **No**. Another dialog box will appear asking whether to Overwrite outlet-temp-avg.out. Click **OK**.

The solution will converge in approximately 35 iterations (Figure 3.12: Residuals for the Coupled Solver Calculation (p. 174)). 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. 175).

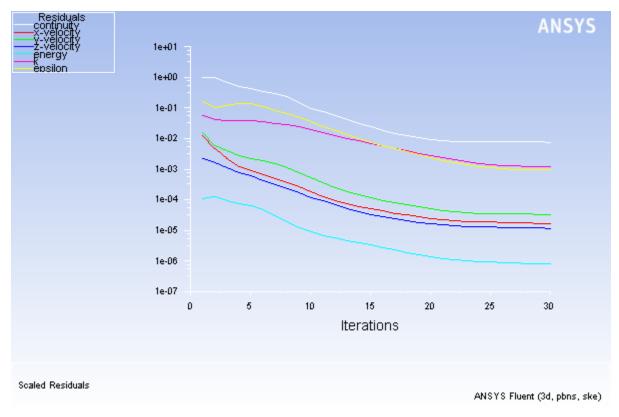


Figure 3.12: Residuals for the Coupled Solver Calculation

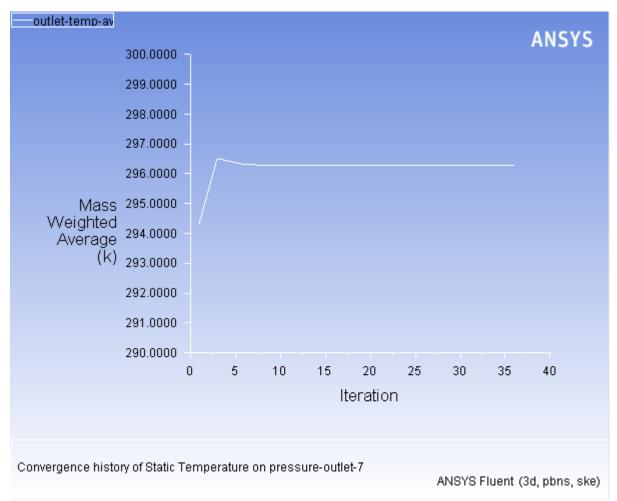


Figure 3.13: Convergence History of Mass-Weighted Average Temperature

3.4.12. 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 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** command that is at the top of the ANSYS Fluent window.

Adapt → Gradient...

Cradient Adaption				-
Options Refine Coarsen Normalize per Zone Contours Manage Controls	Method Curvature Gradient Iso-Value Normalization Scale Normalize Dynamic Interval 20 Curvature Curv	Gradients of Temperature Static Temperature Min 1.421085e-14 Coarsen Threshold 0	Max 0.02833258 Refine Threshold 0.003	•
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 1304 cells were marked for refinement.

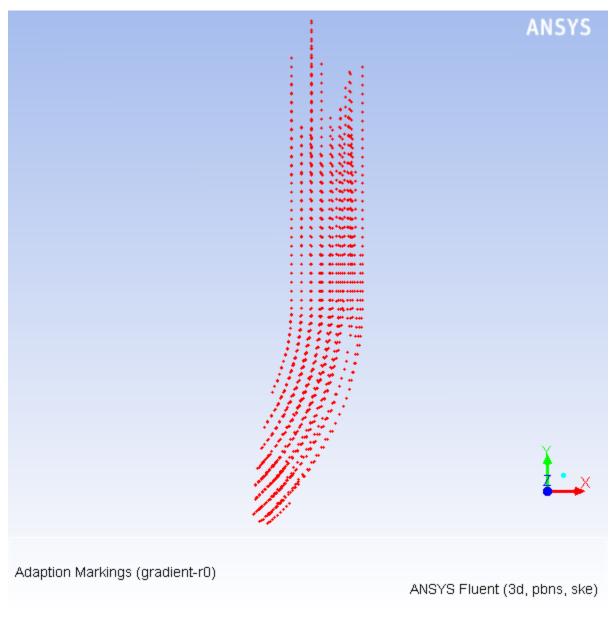
f. Click **Manage...** in the **Gradient Adaptation** panel to open the **Manage Adaption Registers** dialog box.

💶 Manage Adap	tion Registers			×
Register Actions	Registers		Register Info]
Change Type	gradient-r0		gradient-r0	
Combine			Reg ID: 0 Refn #: 1304	
Delete			Crsn #: 0 Type: adapt	
Mark Actions				
Exchange				
Invert				
Limit				
Fill			Options	
			Controls	
Ad	apt Display	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. 178)).





Extra You can change the way ANSYS Fluent displays cells marked for adaption (Figure 3.15: Alternative Display of Cells Marked for Adaption (p. 180)) by performing the following steps:

A. Click **Options...** in the **Manage Adaption Registers** dialog box to open the **Adaption Display Options** dialog box.

💶 Adaption Disp	olay Options	
Options C Draw Mesh Filled	Refine Image: Color red Size Symbol 0.1	Coarsen Wireframe Marker Color Cyan Size Symbol 0.1 ©
	OK Cancel	Help

- B. Enable **Wireframe** in the **Refine** group box.
- C. Enable Filled in the Options group box in the Adaption Display Options dialog box.
- D. Enable **Draw Mesh** in the **Options** group box.

The **Mesh Display** dialog box will open.

💶 Mesh Display			×
Nodes (V Edges (Faces (Partitions	ge Type All Feature Outline	Surfaces default-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall	
0 20 Surface Name Patter	'n	z=0_outlet	
Outline Interior	Match	Surface Types axis clip-surf exhaust-fan fan	
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 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. 180).

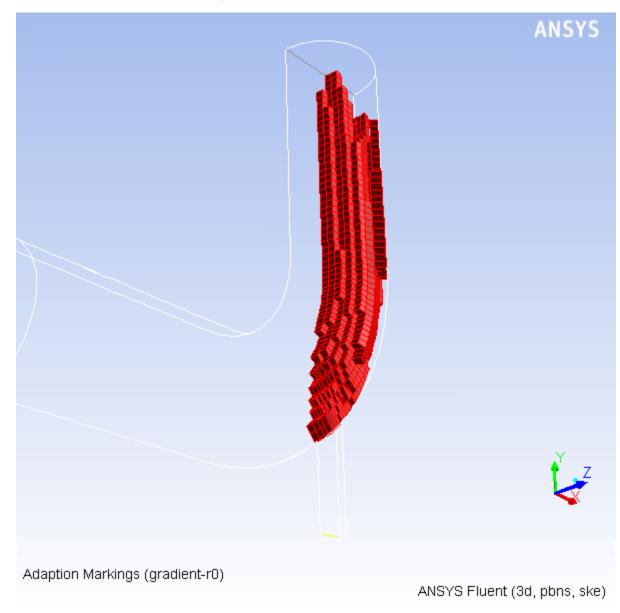


Figure 3.15: Alternative Display of Cells Marked for Adaption

- L. After viewing the marked cells, rotate the view back and zoom out again.
- ii. 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.

Question		×
?	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. 182)).



💶 Mesh Displa	iy		×
Options	Edge Type	Surfaces	
Nodes	All	default-interior	
Edges Edges	 Feature Outline 	pressure-outlet-7 symmetry	
Partitions	Oddine	velocity-inlet-5	
]	velocity-inlet-6	
Shrink Factor	Feature Angle	wall	
0	20	z=0_outlet	
Surface Name Pa	attern Match	New Surface	
	Hater	Surface Types	
Outline Inter	rior	axis	
		clip-surf	
Adjacency		exhaust-fan	
		fan -	*
Display Colors Close Help			

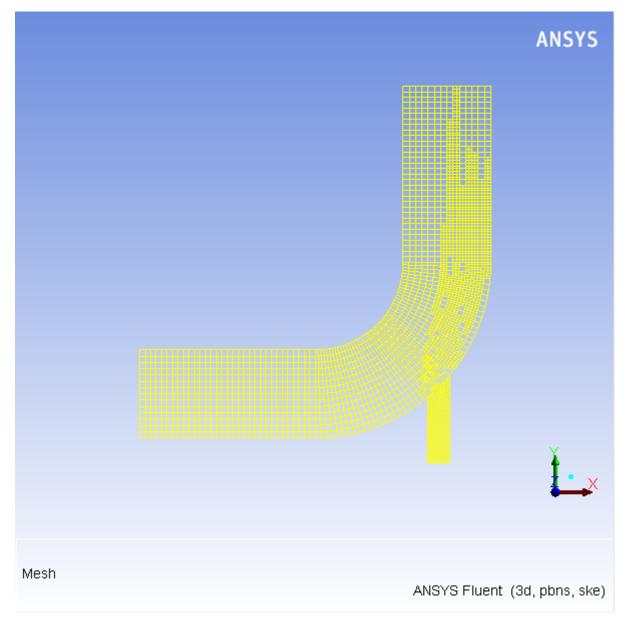
- a. Select All in the Edge Type group box.
- b. Deselect all of the highlighted items from the Surfaces selection list except for symmetry.

Tip

To deselect all surfaces click the far-right unshaded button at the top of the **Surfaces** selection list, and then select the desired surfaces 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.

Run Calculation

Click Calculate.

Run Calculation	
Check Case	Preview Mesh Motion
Number of Iterations	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

The solution will converge after approximately 35 additional iterations (Figure 3.17: The Complete Residual History (p. 183) and Figure 3.18: Convergence History of Mass-Weighted Average Temperature (p. 184)).

ANSY 1e+01 energi èpsilon 1e+00 1e-01 1e-02 1e-03 1e-04 1e-05 1e-06 1e-07 10 20 30 60 ٥ 40 50 Iterations Scaled Residuals ANSYS Fluent (3d, pbns, ske)

Figure 3.17: The Complete Residual History

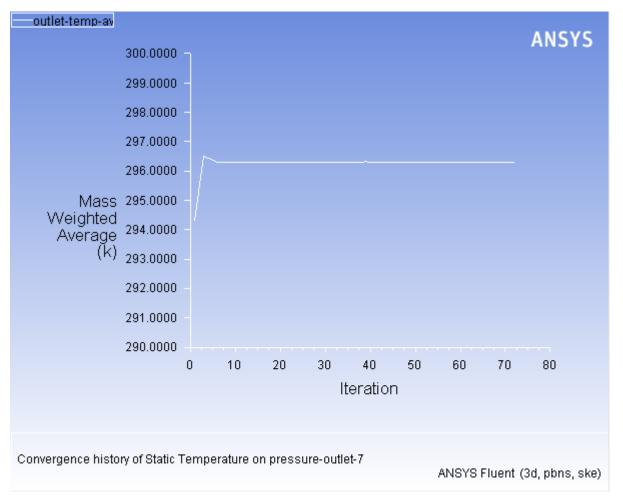


Figure 3.18: Convergence History of Mass-Weighted Average Temperature

4. Save the case and data files for the Coupled solver solution with an adapted mesh (elbow2.cas.gz and elbow2.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

- 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 directory.

5. Examine the filled temperature distribution (using node values) on the revised mesh (Figure 3.19: Filled Contours of Temperature Using the Adapted Mesh (p. 185)).

• Graphics and Animations $\rightarrow \equiv$ Contours \rightarrow Set Up...

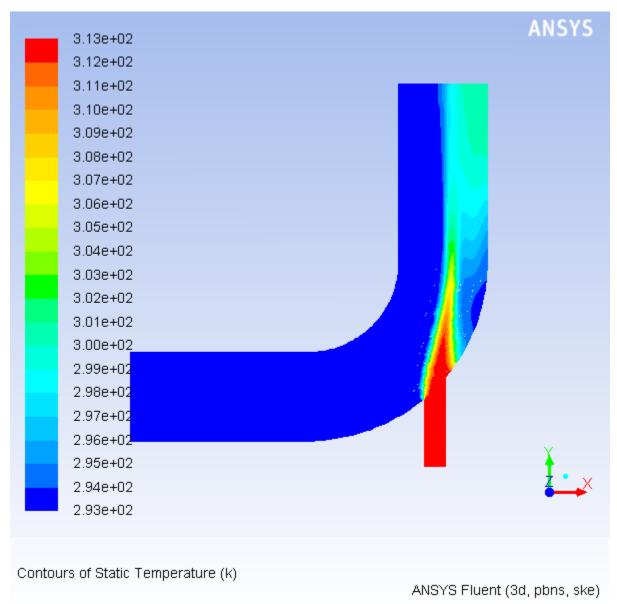


Figure 3.19: Filled Contours of Temperature Using the Adapted Mesh

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. 187)).



Solution XY Plot		
Options Node Values Position on X Axis Position on Y Axis Write to File Order Points	Plot Direction X 1 Y 0 Z 0 Load File	Y Axis Function Temperature Static Temperature X Axis Function Direction Vector Surfaces default-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall z=0_outlet New Surface New Surface
Plot	Axes	Curves Close Help

a. Disable Write to File in the Options group box.

The button that was originally labeled Write... will change to Plot.

- b. Ensure that **Temperature...** and **Static Temperature** are selected from the **Y Axis Function** dropdown lists.
- c. Ensure that **z=0_outlet** is selected from the **Surfaces** selection list.
- d. Click Plot.

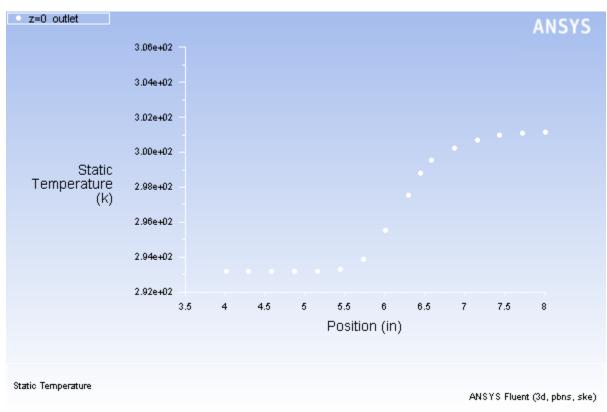


Figure 3.20: Outlet Temperature Profile for the Adapted Coupled Solver Solution

e. Enable Write to File in the Options group box.

The button that was originally labeled **Plot** will change to **Write...**.

- f. Click Write... to open the Select File dialog box.
 - i. 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. 190)).

E File XY Plot		×
Plot Title	Legend Title	
Static Temperature	Static Temperature	
Files	Legend Entries	
C:\Tutorials\introduction\outlet_temp1.xy	Before Adaption	
C:\Tutorials\introduction\outlet_temp2.xy	Adapted Mesh	
		Add
		Delete
C:\Tutorials\introduction\outlet_temp2.xy	Adapted Mesh	Change Legend Entry
Plot Axes	Curves Close Help	

a. Click the **Add...** button to open the **Select File** dialog box.

Select File		—
Look in:	: 🌗 introduction 👻 🌀 🤌 📴 🔻	
Recent Places Desktop Libraries Computer Computer	<pre>elbow.msh elbow1.cas.gz elbow1.dat.gz elbow2.cas.gz elbow2.dat.gz outlet_temp1.xy outlet_temp2.xy outlet-temp-avg.out</pre>	
Network		ОК
	Files of type: XY Files	Cancel
	XY File(s) Remove C:\Tutorials\introduction\outlet_temp1.xy Remove 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 directory in the **XY File(s)** list to indicate that they have been selected.

Tip

If you select a file by mistake, simply click the file in the **XY File(s)** list and then click **Remove**.

- ii. Click **OK** to save the files and close the **Select File** dialog box.
- b. Select the directory path ending in **outlet_temp1.xy** from the **Files** selection list.
- c. Enter Before Adaption in the lowest text-entry box on the right (next to the **Change Legend Entry** button).
- 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 directory 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. 190) 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.

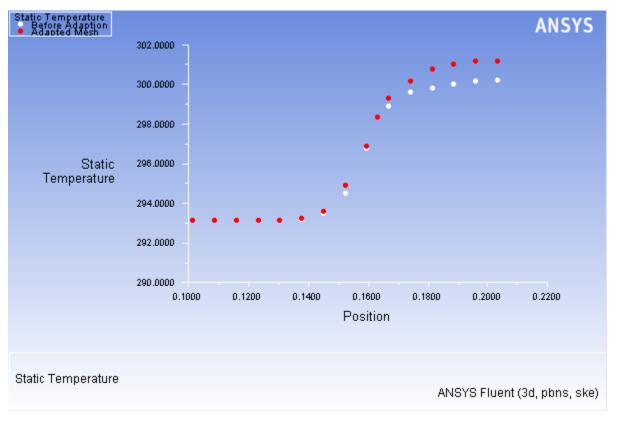


Figure 3.21: Outlet Temperature Profiles for the Two Solutions

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 **Units...** 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. A case and data file (elbow3.cas.gz and elbow3.dat.gz) have been provided in the solution_files directory, in which the mesh has undergone three more levels of adaption. 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. 191).

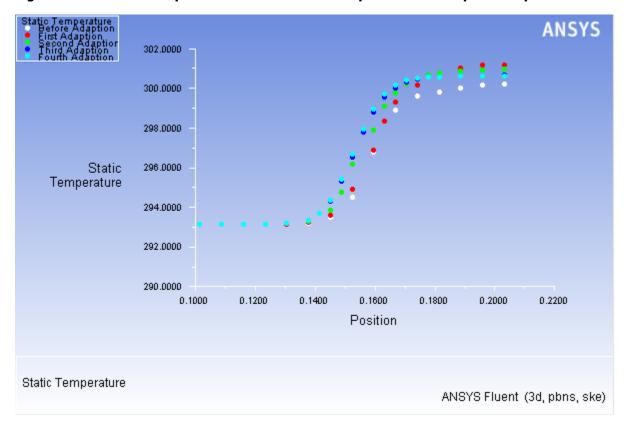


Figure 3.22: Outlet Temperature Profiles for Subsequent Mesh Adaption Steps

It is evident from Figure 3.22: Outlet Temperature Profiles for Subsequent Mesh Adaption Steps (p. 191) 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 300.6 K after mesh independence is achieved. If the adaption steps had not been performed, the wall temperature would have incorrectly been estimated at 299.1 K.

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. 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.

In this problem, the flow field is decoupled from temperature, since all properties are constant. For such cases, it is more efficient to compute the flow-field solution first (that is, without solving the energy equation) and then solve for energy (that is, without solving the flow equations). You will use the **Equations** dialog box to turn the solution of the equations on and off during such a procedure.

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. 123)

and that you are familiar with the ANSYS Fluent navigation pane and menu 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. 194). 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.

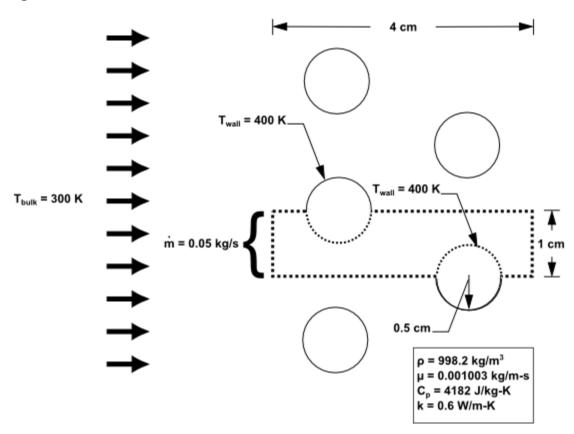


Figure 4.1: Schematic of the Problem

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. 194). 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. 194).

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 15.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip periodic_flow_heat_R150.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 Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, 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.

$\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

2. Check the mesh.

$\mathbf{O}_{General} \rightarrow \mathbf{Check}$

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.

 $\textcircled{} General \rightarrow Scale...$

💶 Scale Mesh	1			—
Domain Extents				 Scaling
Xmin (m) 0		Xmax (m)	0.04	 Convert Units Specify Scaling Factors
Ymin (m) 0		Ymax (m)	0.01	Mesh Was Created In
View Length Uni	it In			Scaling Factors X 0.01 Y 0.01 Scale Unscale
		C	ose Help	

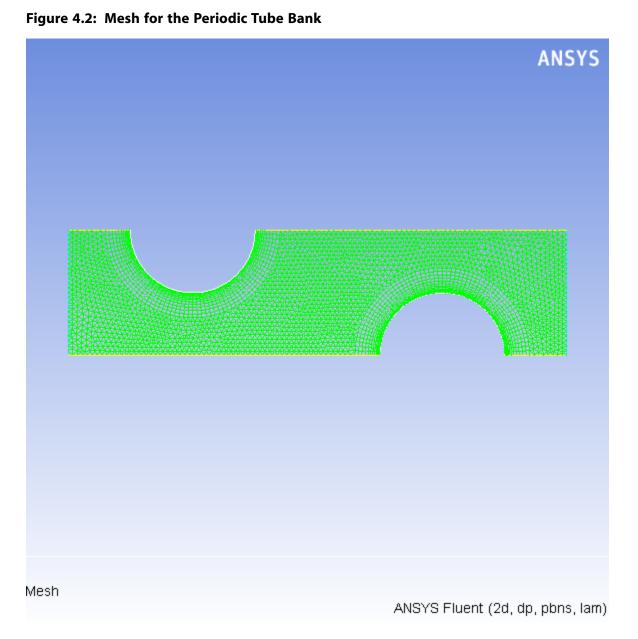
- a. Select **cm** (centimeters) from the **Mesh Was Created In** drop-down list in the **Scaling** group box.
- b. Click **Scale** to scale the mesh.
- c. Close the **Scale Mesh** dialog box.
- 4. Check the mesh.

 $\mathbf{O}_{\mathsf{General}} \rightarrow \mathsf{Check}$

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. 197)).



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. 197)). 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.

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.

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 outlined in boxes as shown:

```
mesh/modify-zones/make-periodic
```

```
Periodic zone [()] 9
Shadow zone [()] 12
Rotational periodic? (if no, translational) [yes] no
Create periodic zones? [yes] yes
Auto detect translation vector? [yes] yes
```

zone 12 deleted

created periodic zones.

4.4.3. General Settings

1. Retain the default settings for the solver.

O	Gan	eral
•	Gen	erai

General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative
Time ◉ Steady ◯ Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

4.4.4. Models

1. Enable heat transfer.

```
 \mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Energy} \to \mathsf{Edit...}
```

💶 Energy 🛛 💽		
Energy		
Energy Equation		
	-	
OK Cancel Help		

- a. Enable Energy Equation.
- b. Click **OK** to close the **Energy** dialog box.

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.

 $\clubsuit Materials \rightarrow \blacksquare Fluid \rightarrow Create/Edit...$

a. Click **Fluent Database...** in the **Create/Edit Materials** dialog box to open the **Fluent Database Ma-terials** dialog box.

FLUENT Database Materials				×
FLUENT Fluid Materials vinyl-silylidene (h2cchsih) vinyl-trichlorosilane (sid3ch2ch) vinylidene-chloride (ch2cd2) water-liquid (h2o <l>) water-vapor (h2o) wood-volatiles (wood_vol) (</l>		Material Type fluid Order Materials by Name Chemical Formula		
Properties				
Density (kg/m3)	constant 998.2		▼ View	•
Cp (Specific Heat) (j/kg-k)	constant 4182		▼ View	
Thermal Conductivity (w/m-k)	constant		View	
Viscosity (kg/m-s)	constant 0.001003		View	Ŧ
New Edit	Save	Copy Close	Help	

i. Select water-liquid (h2o<l>) in the Fluent Fluid Materials selection list.

Scroll down the list to find **water-liquid (h2o<l>)**. 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>)</l>		 FLUENT Database
	Mixture		User-Defined Database
	none		Ŧ
Properties			
Density (kg/m3)	constant) â	
	998.2		
Cp (Specific Heat) (j/kg-k)	constant 💌 Edit		
	4182		
Thermal Conductivity (w/m-k)	constant		
	0.6		
Viscosity (kg/m-s)	constant 👻 Edit		
	0.001003	- U	

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**).

♦ Cell Zone Conditions $\rightarrow \stackrel{\frown}{=}$ fluid-16 \rightarrow Edit...

E Fluid
Zone Name
fluid-16
Material Name water-liquid
Frame Motion Source Terms Mesh Motion Fixed Values
Porous Zone
Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Multiphase
Rotation-Axis Origin X (m) 0 constant
OK Cancel Help

- 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.

 $\textcircled{P} Boundary Conditions \rightarrow \fbox{periodic-9} \rightarrow \texttt{Periodic Conditions...}$

Periodic Conditions	—
Type Specify Mass Flow Specify Pressure Gradient Mass Flow Rate (kg/s) 0.05 Pressure Gradient (pascal/m) 0 Upstream Bulk Temperature (k) 300	Flow Direction X 1 Y 0 Z 0 Relaxation Factor 0.5 Number of Iterations 2 2 •
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).

 $\textcircled{P} Boundary Conditions \rightarrow \fbox{wall-21} \rightarrow \texttt{Edit...}$

🔁 Wall			•••
Zone Name			
wall-bottom			
Adjacent Cell Zone			
fluid-16			
Momentum Thermal Rad	diation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
Heat Flux	Temperature (k) 400	constant 🔹
 Temperature Convection 		Wall Thickness	(m) 0
Radiation			
 Mixed via System Coupling 	Heat Generation Rate (w/m	³⁾ 0	constant
Material Name			
aluminum	▼ Edit		
	OK	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).

Gradient Conditions → 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.

✤Solution 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	
· · · · · · · · · · · · · · · · · · ·	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
V Pseudo Transient	
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 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. Use of this option is recommended for general fluid flow problems.

2. Set the solution controls.

Solution Controls

Solution Controls

Pressure	^
0.5	
Momentum	
0.5	
Density	
1	
Body Forces	
1	
Energy	
0.75	
Default	
Equations Limits Advanced	
Help	

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 Setting Pseudo Transient Explicit Relaxation Factors in the Fluent User's Guide.

3. Enable the plotting of residuals during the calculation.

options	Equations				
✓ Print to Console	Residual	Monitor C	heck Convergen	ce Absolute Criteria	~
V Plot	continuity			0.001	
1 Curves Axe	x-velocity	V		0.001	
Iterations to Plot	y-velocity	V		0.001	
1000	energy			1e-06	-
	Residual Values			Convergence Cr	iterion
1000	🕅 Normalize		Iterations	absolute	
	Scale				
	Compute Lo	cal Scale			

- a. Ensure **Plot** is enabled in the **Options** group box.
- b. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Initialize the solution.

Solution Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
More Settings Initialize
Patch
Reset DPM Sources Reset Statistics
Help

a. Retain the default selection of **Hybrid Initialization** in the **Initialization Methods** group box.

b. Click Initialize.

c. 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.

Solution Initialization → Patch...

Patch		×
Reference Frame Relative to Cell Zone Absolute Variable Pressure X Velocity Y Velocity Temperature	Value (k) 300 Use Field Function Field Function	Zones to Patch
	Patch Close 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).

 $\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}$

6. Start the calculation by requesting 350 iterations.

Run Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Pseudo Transient Options	
Fluid Zone	
Time Step Method	Timescale Factor
 User Specified Automatic 	1
Length Scale Method	Verbosity
Conservative -	
Number of Iterations	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- a. Enter 350 for Number 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).

File \rightarrow Write \rightarrow Case & Data...

4.4.10. Postprocessing

1. Display filled contours of static pressure (Figure 4.3: Contours of Static Pressure (p. 210)).

Contours	×
Options	Contours of
V Filled	Pressure
✓ Node Values ✓ Global Range	Static Pressure 🔻
Auto Range Clip to Range	Min Max
Draw Profiles	
Draw Mesh	Surfaces 🗵 🗏 🗏
	interior-15
	periodic-9
Levels Setup	symmetry-11
20 🔺 1 🔺	symmetry-13
	symmetry-18 👻
Surface Name Pattern	New Surface 🔻
Match	Surface Types
	axis
	clip-surf
	exhaust-fan
	fan 👻
	- ·
Display	Compute Close Help

\bigcirc Graphics and Animations → **\sqsubseteq** Contours → Set Up...

- 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**.

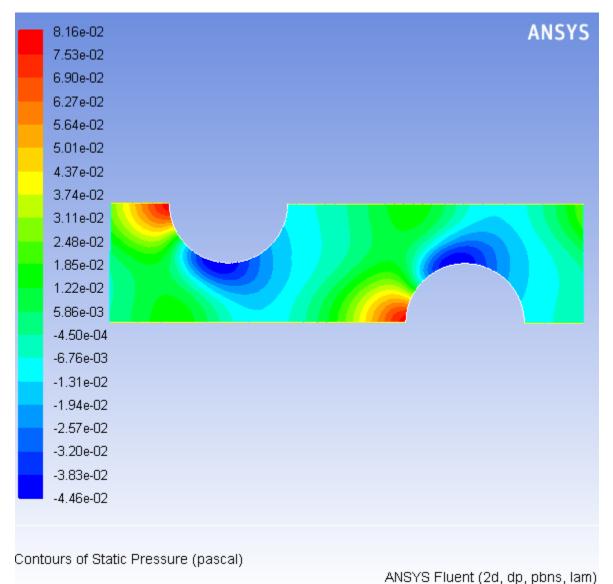


Figure 4.3: Contours of Static Pressure

d. Change the view to mirror the display across the symmetry planes (Figure 4.4: Contours of Static Pressure with Symmetry (p. 212)).

Graphics and Animations → Views...

U Views		×
Views back bottom front isometric left right top Save Name view-0	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes 🖹 🖹 🗐 symmetry-18 symmetry-13 symmetry-11 symmetry-24 Define Plane Periodic Repeats Define
Apply Came	ra Close	Help

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 comprised 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. 212) 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. 212)).

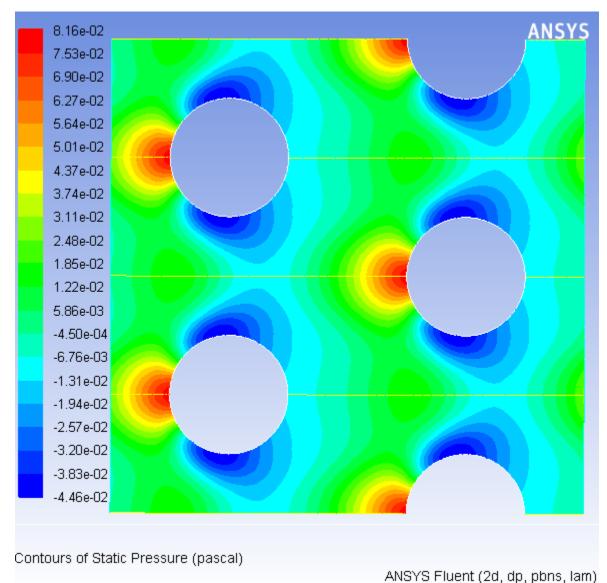


Figure 4.4: Contours of Static Pressure with Symmetry

The pressure contours displayed in Figure 4.4: Contours of Static Pressure with Symmetry (p. 212) 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. 214)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

Contours	—
Options	Contours of
V Filled	Temperature 👻
 Node Values Global Range 	Static Temperature
Auto Range	Min (k) Max (k)
Clip to Range	277.0558 400
Draw Profiles	
Draw Mesh	Surfaces
	interior-15
Levels Setup	periodic-9
	symmetry-11
20 🔺 1	symmetry-13
	symmetry-18 👻
Surface Name Pattern	New Surface
Match	Surface Types
	axis
	clip-surf
	exhaust-fan
	fan 👻
	" - ·
Display	Compute Close Help

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

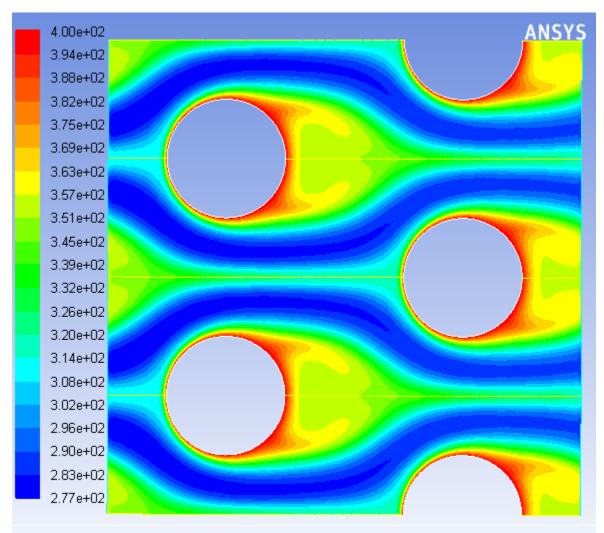


Figure 4.5: Contours of Static Temperature

Contours of Static Temperature (k)

ANSYS Fluent (2d, dp, pbns, lam)

The contours in Figure 4.5: Contours of Static Temperature (p. 214) 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. 216)).

Graphics and Animations → \blacksquare Vectors → Set Up...

.				
Vectors			×	
Options	Vectors of			
🔽 Global Range	Velocity		•	
Auto Range	Color by			
Clip to Range	Velocity			
Draw Mesh	Velocity Magnitude		•	
Style	Min (m/s)	Max (m/s)		
arrow	1.647334e-06	0.01312426		
Scale Skip	Surfaces			
2 0	interior-15		*	
Vector Options	periodic-9		=	
vector options	symmetry-11		-	
Custom Vectors	symmetry-13 symmetry-18			
	symmetry-24			
Surface Name Pattern			*	
Match	New Surface 🔻			
	Surface Types			
	axis			
	clip-surf			
	exhaust-fan			
	fan		*	
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. 216)), 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. 216) clearly shows the recirculating flow behind the tube and the boundary layer development along the tube surface.

	* • • • • • • • • • • • • • • • • • • •		• • • • • •	br br br p	* 8 * 8 * 8 * 8 * 8 * 8
-	1.31e-02				<mark>⊳ ► ↓ ► ↓</mark> ANSYS
	1.25e-02 P + + + + + + + + + + + + + + + + + +	॑₽₽₽₽₽ ₽			
			₽₽ ₽₽₽₽₽	₽_₽₽₿₽₿₽	
			₽_₽₽_₽₽_₽₽	₽₽₽₽₽₽	ÈşÈşÈşÈşÈş Èş
			₽₽₽₽₽₽ ₽	₽₽₽₽₽ ₽	<u>, e, e, e, e, e, e</u> , e,
2		P P P P P P	₽_₽₽<u>₽</u>₽₽₽₽ ₽	<u>FREE FE</u>	i de la d
	9846.03	Ĩ₽₽₽₽ ₽₽₽	, ₽_₽₽_₽₽ ₽	⋭⋤⋭<u></u>⋤⋭	
	9 19e 103 - P P P	₽ <mark>₽₽₽₽₽₽</mark> ₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽₽		, 1 , 1, 1, 1,	
	27:88e-03				to to to to the
	.7,22e-03 · · · · · · · · ·				
	6.56e-03				
	5:9fe-03				and the states
	5.25e-03				and a state of the
		a se je	1,-721		
	4.59e-03		T Z j	.=_= = =	Z-Z-Z
	3.94e-03	Charles and		92.F.F.	
	3.28e-03	$\propto e^{-1}e^$	al de la	1.1.1.2.2	22222
	2.63e-03	$\sim \times 10^{-1}$	Start's	Sugar, Cal	行われていてい
	1.97e-03	and the second sec	e^{2} , e^{2} ,	526 C. C.	
		and the second sec	$\phi_{ij}^{(1)}$ ($\phi_{ij}^{(1)}$	estates.	A State of the second
	1.31e-03	\.·	all and a	1. S. 2. A.	Sector Sector
	6.58e-04		and the second	1. S. S. S. S.	Server
	1.65e-06		Sec. Sec.		nanan esere e
			1000	1. 1. 2. 2	and the second second

Figure 4.6: Velocity Vectors

Velocity Vectors Colored By Velocity Magnitude (m/s)

ANSYS Fluent (2d, dp, pbns, lam)

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.

Surface \rightarrow Iso-Surface...

Iso-Surface		×
Surface of Constant Mesh X-Coordinate Min (m) Max (m) 0 Iso-Values (m) 0.01 New Surface Name x=0.01m	From Surface interior-15 periodic-9 symmetry-11 symmetry-13 symmetry-18 symmetry-24 Trom Zones fluid-16	
Create Compute Manage.	Close Help	

- 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. 219)).

♦ Plots $\rightarrow \equiv XY$ Plot \rightarrow Set Up...

Solution XY Plot		
Options Image: Options Image: Option on X Axis Image: Position on Y Axis Image: Order Points	Plot Direction X 0 Y 1 Z 0 Load File Free Data	Y Axis Function Temperature Static Temperature X Axis Function Direction Vector Surfaces Symmetry-18 symmetry-24 wall-bottom wall-top x=0.01m x=0.02m x=0.03m Vew Surface Vector
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 -	Solution XY Plot	—	
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.

ANSYS 4.00e+02 3.80e+02 3.60e+02 3.40e+02 Static Temperature 3.20e+02 (k) 3.00e+02 2.80e+02 2.60e+02 ٥ 0.001 0.002 0.003 0.004 0.005 0.006 0.007 0.008 0.009 0.01 Position (m) Static Temperature

Figure 4.7: Static Temperature at x=0.01, 0.02, and 0.03 m

ANSYS Fluent (2d, dp, pbns, lam)

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 Modeling Periodic Heat Transfer in 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. 123).

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 monitors 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:

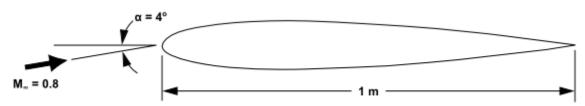
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

and that you are familiar with the ANSYS Fluent navigation pane and menu 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. 222).

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. General Settings
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 **15.0** under **Version**.
- 5. Select this tutorial from the list.

- 6. Click **Files** to download the input and solution files.
- 7. Unzip external_compressible_R150.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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, 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.

```
\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}
```

```
2. Check the mesh.
```

```
    General → Check
```

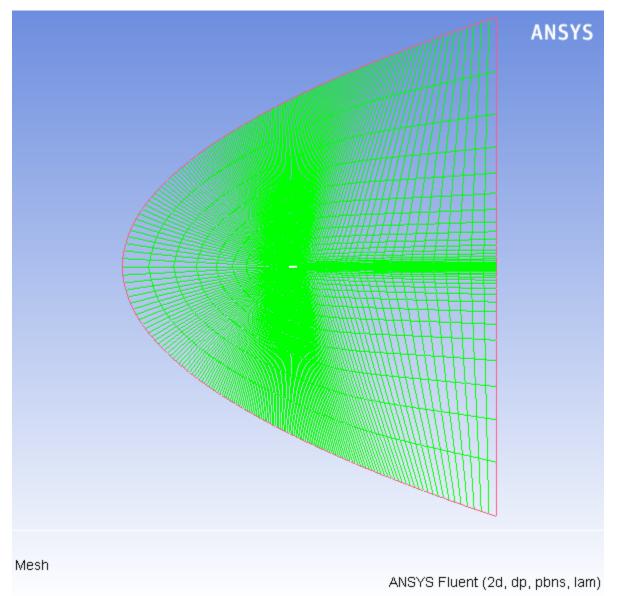
ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Make sure that the reported minimum volume is a positive number.

Note

ANSYS Fluent will issue a warning concerning the high aspect ratios of some cells and possible impacts on calculation of Cell Wall Distance. The warning message includes recommendations for verifying and correcting the Cell Wall Distance calculation. In this particular case the cell aspect ratio does not cause problems so no further action is required. As an optional activity, you can confirm this yourself after the solution is generated by plotting Cell Wall Distance as noted in the warning message.

3. Examine the mesh (Figure 5.2: The Entire Mesh (p. 224) and Figure 5.3: Magnified View of the Mesh Around the Airfoil (p. 225)).

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 comprised of quadrilateral and triangular cells.

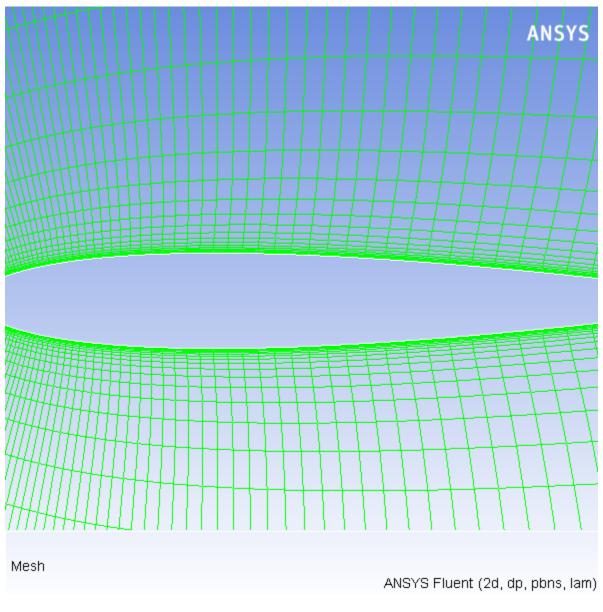


Figure 5.3: Magnified View of the Mesh Around the Airfoil

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.

4. Reorder the mesh.

 $\textbf{Mesh} \rightarrow \textbf{Reorder} \rightarrow \textbf{Domain}$

This is done to reduce the bandwidth of the cell neighbor number and to speed up the computations. This is especially important for large cases involving 1 million or more cells. The method used to reorder the domain is the Reverse Cuthill-McKee method.

5.4.3. General Settings

1. Set the solver settings.

General	
Mesh Scale Display	Check Report Quality
Solver	
Type ● Pressure-Based ○ Density-Based	Velocity Formulation Absolute Relative
Time ◉ Steady ◯ Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

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.

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 (5 eqn) Scale-Adaptive Simulation (SAS) Spalart-Allmaras Production Vorticity-Based Strain/Vorticity-Based Strain/Vorticity-Based Coptions Curvature Correction	Model Constants Cb1 (1355) (Cb2) (Cb2) (Cv1) (7.1) (Cw2) (0.3) User-Defined Functions User-Defined Functions Turbulent Viscosity none ((((
ОК	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.

• Materials $\rightarrow \equiv$ air \rightarrow Create/Edit...

Create/Edit Materials					
Name air		Material Type		-	Order Materials by
Chemical Formula		FLUENT Fluid Materia	ıls		Chemical Formula
		Mixture		•	User-Defined Database
Properties		none			
Density (kg/m3)	ideal-gas		• Edit	Î	
Cp (Specific Heat) (j/kg-k)	constant 1006.43		► Edit	E	
Thermal Conductivity (w/m-k)	constant		• Edit		
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	×
Methods	
 Two Coefficient Method (SI Units Only) Three Coefficient Method 	
Reference Viscosity, mu0 (kg/m-s) 1.716e-05	*
Reference Temperature, T0 (k) 273.11	
Effective Temperature, S (k) 110.56	Ŧ
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

Order Boundary Conditions

Boundary Cor	nditions	
Zone		
interior-1 pressure-far-field wall-bottom wall-top	-1	
Phase mixture	Type ▼ pressure-far-field ▼	ID 11
Edit Parameters Display Mesh	Copy Profiles Operating Conditions Periodic Conditions	
Help		

1. Set the boundary conditions for **pressure-far-field-1**.

 $\textcircled{P} Boundary Conditions \rightarrow \fbox{Pressure-far-field-1} \rightarrow \texttt{Edit...}$

Pressure Far-Field				
Zone Name	Zone Name			
pressure-far-field-1				
Momentum Thermal Radiation	on Species 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	Furbulent Viscosity Ratio	•		
Turbulent Viscosity Ratio	10	constant 👻		
OK Cancel 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 UDS DPM			
Temperature (k) 300 constant			
OK Cancel Help			

g. Click **OK** to close the **Pressure Far-Field** dialog box.

5.4.7. Operating Conditions

1. Set the operating pressure.

 $\clubsuit Boundary \ Conditions \rightarrow Operating \ Conditions...$

Operating Conditions	— ×
Pressure	Gravity
Operating Pressure (pascal)	Gravity
101325 P	
Reference Pressure Location	
X (m) 0	
Y (m) 0	
Z (m) 0	
OK Cancel Hel;	2

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 Operating Pressure in the User's Guide.

5.4.8. Solution

1. Set the solution parameters.

Solution Methods

Solution Methods		
Pressure-Velocity Coupling		
Scheme		
Coupled	-	
Spatial Discretization		
Gradient		Ê
Least Squares Cell Based	-	
Pressure		
Second Order	-	Ξ
Density		=
Second Order Upwind		
Momentum		
Second Order Upwind		-
Modified Turbulent Viscosity		
Second Order Upwind	-	-
Transient Formulation		
	-	
Non-Iterative Time Advancement		
Frozen Flux Formulation		
Pseudo Transient		
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 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.

Colution Controls

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Density	^
0.5	
Body Forces	
1	h
Modified Turbulent Viscosity	
0.9	
Turbulent Viscosity	Ξ
1	
Energy	
0.75	
J. T. J.	Ŧ
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.

Monitors → Eresiduals → Edit...

Residual Monitors					x
Options	Equations				-
Print to Console	Residual	Monitor (Check Convergen	ce Absolute Criteria	
Vindow I Curves Iterations to Plot 1000	continuity	\checkmark	\checkmark	0.001	
	x-velocity	V		0.001	E
	y-velocity	\checkmark		0.001	
	energy	V		1e-06	-
	Residual Values			Convergence Cr	iterion
Iterations to Store	Normalize		Iterations	absolute	•
	Scale	al Scale			
OK Plot Renormalize Cancel Help					

- 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 Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
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]
 Number of FMG (and FAS geometric multigrid) levels: 5
 * FMG customization summary:
    residual reduction on level 0 [finest grid] is: 0.001
    number of cycles on level 0 is: 1
    residual reduction on level 1 is: 0.001
    number of cycles on level 1 is: 100
    residual reduction on level 2 is: 0.001
    number of cycles on level 2 is: 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 Full Multigrid (FMG) Initialization in the User's Guide.

5. Save the case and data files (airfoil.cas.gz and airfoil.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

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.

CRun Calculation

- a. Enter 50 for Number of Iterations.
- b. Click Calculate.

By performing some iterations before setting up the force monitors, you will avoid large initial transients in the monitor 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.

Reference Values

The reference values are used to nondimensionalize 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)	277.6701
Viscosity (kg/m-s)	1.7894e-05
Ratio of Specific Heats	1.4
Reference Zone	
	•
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. Define a force monitor to plot and write the drag coefficient for the walls of the airfoil.

♣ Monitors (Residuals, Stastistic and Force Monitors) → Create → $\stackrel{\bullet}{\equiv}$ Drag...

💶 Drag Monitor		— ×-
Name	Wall Zones	
cd-1	wall-bottom	
Options	wall-top	
Print to Console		
V Plot		
Window		
2 Curves Axes		
Write		
File Name		
cd-1-history		
Per Zone		
Average Over(Iterations)		
Force Vector		
X Y Z]
0.9976 0.06976 0		
Save Output Parameter		
OK Plot Clear	Cancel Help	

- a. Enable **Plot** in the **Options** group box.
- b. Enable Write to save the monitor history to a file.

Note

If you do not enable the **Write** option, the history information will be lost when you exit ANSYS Fluent.

- c. Retain the default entry of cd-1-history for File Name.
- d. Select wall-bottom and wall-top in the Wall Zones selection list.
- e. 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° off of the global coordinates.

f. Click **OK** to close the **Drag Monitor** dialog box.

9. Similarly, define a force monitor for the lift coefficient.

� Monitors →	Create →	Lift
--------------	----------	------

Lift Monitor	×
Name	Wall Zones 🗈 🔳 🚍
d-1	wall-bottom
Options	wall-top
Print to Console	
V Plot	
Window 3 Curves Axes	
Vrite	
File Name	
d-1-history	
Per Zone	
Average Over(Iterations)	
Force Vector	
X Y Z -0.0698 0.9976 0	
Save Output Parameter	
OK Plot Clear	Cancel Help

Enter the values for **X** and **Y** shown in the **Lift Monitor** dialog box.

The **X** and **Y** values shown ensure that the lift coefficient is calculated normal to the free-stream flow, which is 4° off of the global coordinates.

10. In a similar manner, define a force monitor for the moment coefficient.

 $\diamondsuit Monitors \rightarrow Create \rightarrow \blacksquare Moment...$

Moment Monitor		×
Name	Wall Zones	
cm-1	wall-bottom	
Options	wall-top	
Print to Console		
V Plot		
4 Curves Axes		
Vrite		
File Name		
cm-1-history		
Per Zone		
Average Over(Iterations)		
1		
Moment Center		
X (m) Y (m) Z (m)		
0.25 0 0		
Moment Axis	_	
X Y Z 0 0 1		
Save Output Parameter		
OK Plot Clear	Cancel Help	

Enter the values for the **Moment Center** and **Moment Axis** shown in the **Moment Monitor** dialog box.

11. Display filled contours of pressure overlaid with the mesh in preparation for defining a surface monitor (Figure 5.4: Pressure Contours After 50 Iterations (p. 241) and Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells (p. 242)).

Contours		×	
Options	Contours of		
Filled	Pressure	•	
V Node Values	Static Pressure	•	
Global Range Auto Range	Min Max		
Clip to Range	0		
🔽 Draw Mesh	Surfaces		
	interior-1		
Levels Setup	pressure-far-field-1 wall-bottom		
20 🔺 1	wall-bottom wall-top		
	Wai-top		
Surface Name Pattern	New Surface 🕶		
Mat	ch Surface Types		
	axis		
	clip-surf		
	exhaust-fan		
	fan	-	
Display	Compute Close Help		

- a. Enable **Filled** in the **Options** group box.
- b. Enable **Draw Mesh** to open the **Mesh Display** dialog box.

💶 Mesh Displa	у		— X—
Options Nodes Edges Faces Partitions	Edge Type All Feature Outline	Surfaces interior-1 pressure-far-field-1 wall-bottom wall-top	
Shrink Factor	Feature Angle		
	Match	New Surface Surface Types axis dip-surf exhaust-fan fan	
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.

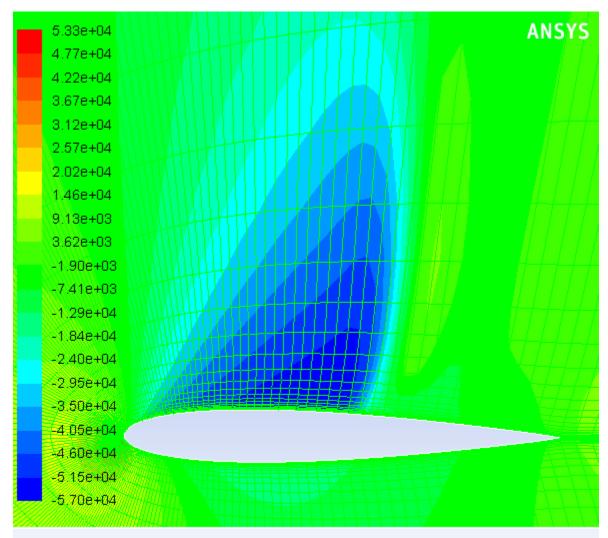


Figure 5.4: Pressure Contours After 50 Iterations

Contours of Static Pressure (pascal)

ANSYS Fluent (2d, dp, pbns, S-A)

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. 242).

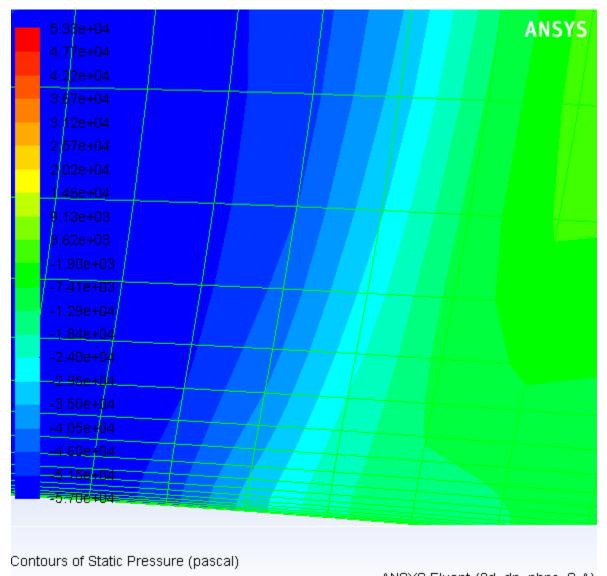


Figure 5.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells

ANSYS Fluent (2d, dp, pbns, S-A)

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 define a surface monitor.

12. Create a point surface just downstream of the shock wave.

Surface → Point...

Point Surface	
Options Point Tool Reset	Coordinates x0 (m) 0.53 y0 (m) 0.051 z0 (m) 0
5	Gelect Point with Mouse
New Surface Nam point-4	e
Create	anage 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. 244)).
- c. Click the right mouse button.
- d. Click **Create** to create the point surface and then close the **Point Surface** dialog box.

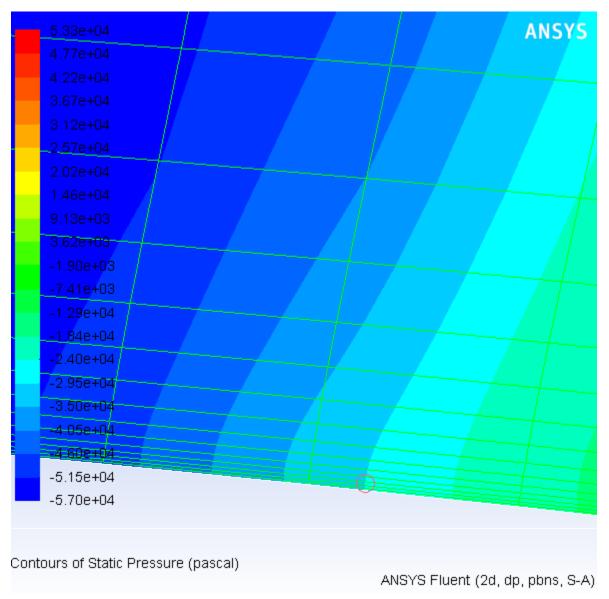


Figure 5.6: Pressure Contours after Creating a Point with the Mouse

13. Enable residual plotting during the calculation.

 $\textcircled{} Monitors \rightarrow \overleftarrow{\blacksquare} Residuals \rightarrow Edit...$

Residual Monitors			
Options Print to Console Plot Window 1 Curves Axes Iterations to Plot 1000	Equations Residual Monitor continuity V x-velocity V energy V		
Iterations to Store	Residual Values Normalize Scale Compute Local Scale	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 so that automatic convergence checking does not occur.
- c. Click **OK** to close the **Residual Monitors** dialog box.
- 14. Define a surface monitor 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, define a monitor at a point (just downstream of the shock) where there is likely to be significant variation, and monitor the value of the velocity magnitude.

♦ Monitors (Surface Monitors) → Create...

Surface Monitor	×
Name	Report Type
surf-mon-1	Vertex Average 🔹
Options	Field Variable
Print to Console	Velocity
V Plot	Velocity Magnitude
Window	Surfaces
5 Curves Axes	interior-1
	point-4
Vrite	pressure-far-field-1
File Name	wall-bottom
surf-mon-1.out	wall-top
M Anto	
X Axis	
Iteration 🔻	
Get Data Every	
1 Iteration V	
Average Over	New Surface -
1	
ОК	Cancel Help

- a. Enable **Plot** and **Write**.
- b. Select Vertex Average from the Report Type drop-down list.

Scroll down the **Report Type** drop-down list to find **Vertex Average**.

- c. Select Velocity... and Velocity Magnitude from the Field Variable drop-down list.
- d. Select point-4 in the Surfaces selection list.
- e. Click **OK** to close the **Surface Monitor** dialog box.

15. Save the case and data files (airfoil-1.cas.gz and airfoil-1.dat.gz).

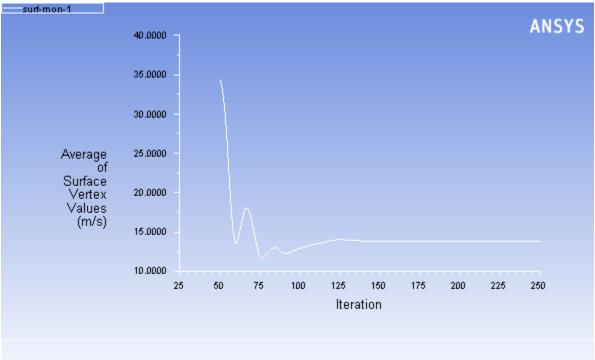
File \rightarrow Write \rightarrow Case & Data...

16. Continue the calculation for 200 more iterations.

Run Calculation

The force monitors (Figure 5.8: Drag Coefficient Convergence History (p. 247) and Figure 5.9: Lift Coefficient Convergence History (p. 248)) show that the case is converged after approximately 200 iterations.





Convergence history of Velocity Magnitude on point-4

ANSYS Fluent (2d, dp, pbns, S-A)

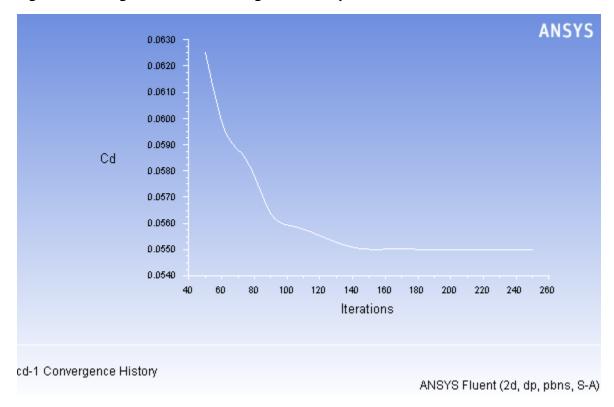
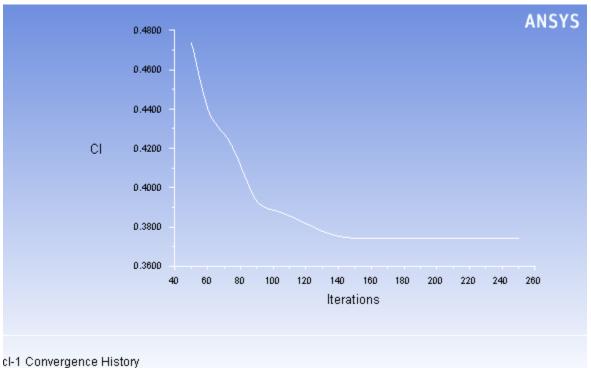


Figure 5.8: Drag Coefficient Convergence History

Figure 5.9: Lift Coefficient Convergence History



ANSYS Fluent (2d, dp, pbns, S-A)

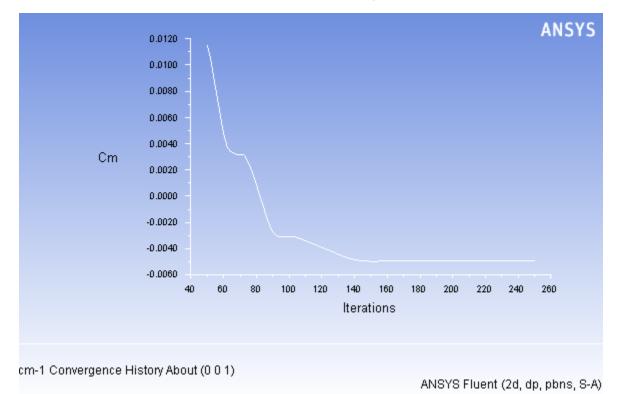


Figure 5.10: Moment Coefficient Convergence History

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. 250)).

Solution XY Plot		
Options Node Values Position on X Axis Position on Y Axis Write to File Order Points	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Turbulence Wall Yplus X Axis Function Direction Vector Surfaces INTERIOR
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

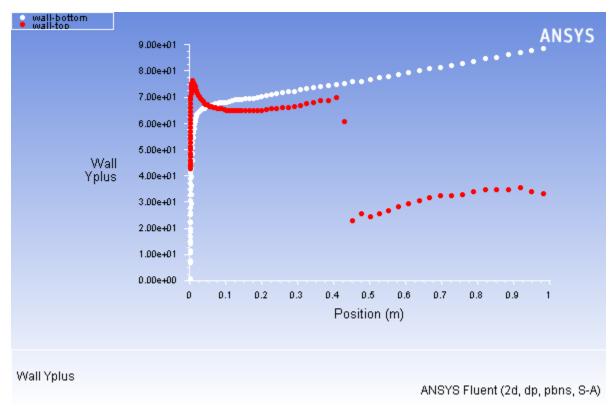
The values of y^+ are dependent on the resolution of the mesh and the Reynolds number of the flow, and are defined only in wall-adjacent cells. The value of y^+ in the wall-adjacent cells dictates how wall shear stress is calculated. When you use the Spalart-Allmaras model, you should check that y^+ 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, μ is the molecular viscosity, ρ is the density of the air, and τ_w is the wall shear stress.

Figure 5.11: XY Plot of y+ Distribution (p. 250) 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.

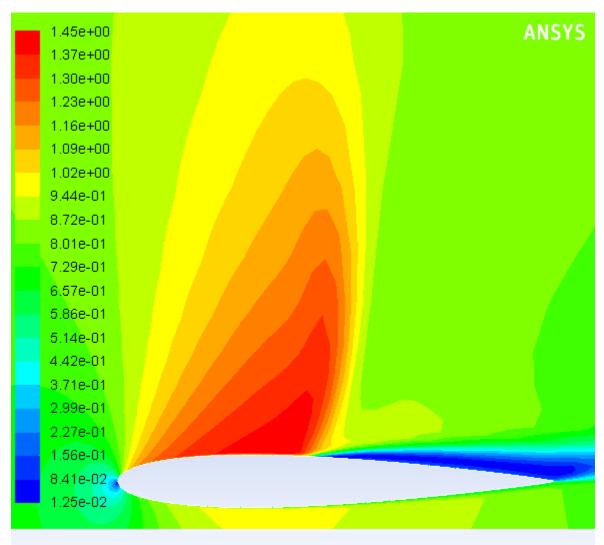




2. Display filled contours of Mach number (Figure 5.12: Contour Plot of Mach Number (p. 251)).

\bigcirc Graphics and Animations $\rightarrow \equiv$ Contours \rightarrow Set Up...

- 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. 251).





Contours of Mach Number

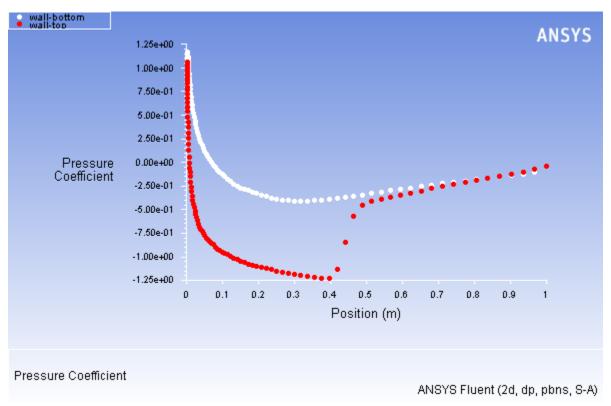
ANSYS Fluent (2d, dp, pbns, S-A)

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. 251) at about $x/c \approx 0.45$.

3. Plot the pressure distribution on the airfoil (Figure 5.13: XY Plot of Pressure (p. 252)).

- 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. 252).

- 4. Plot the *x* component of wall shear stress on the airfoil surface (Figure 5.14: XY Plot of x Wall Shear Stress (p. 253)).
 - 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. 253), 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.

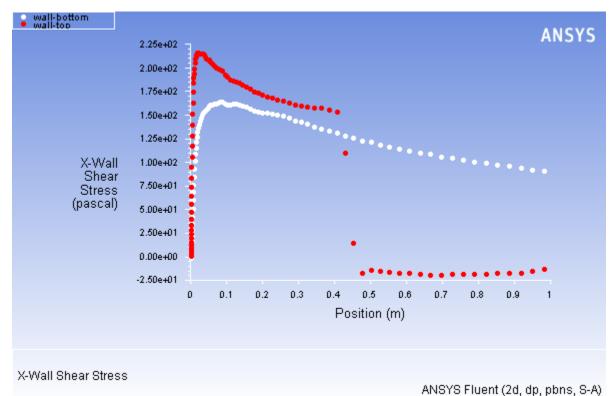


Figure 5.14: XY Plot of x Wall Shear Stress

5. Display filled contours of the *x* component of velocity (Figure 5.15: Contour Plot of x Component of Velocity (p. 254)).



- a. Ensure Filled is enabled in the Options group box.
- b. Select Velocity... and X Velocity from the Contours of drop-down lists.

Scroll up in the **Contours of** drop-down list to find **X Velocity**.

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. 254).

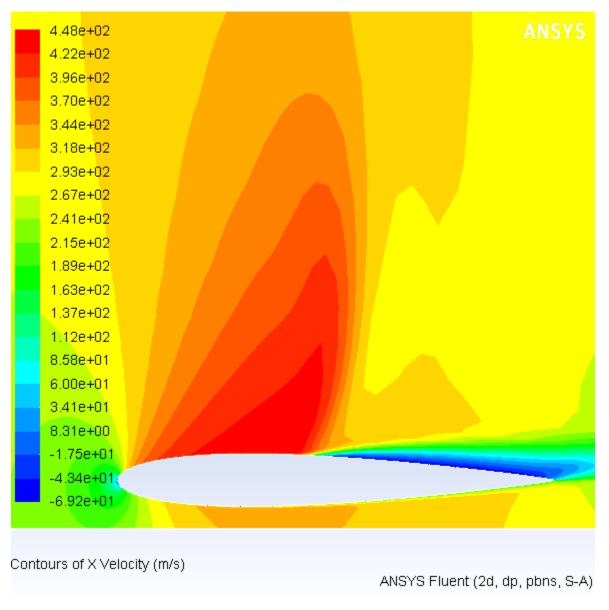


Figure 5.15: Contour Plot of x Component of Velocity

6. Plot velocity vectors (Figure 5.16: Plot of Velocity Vectors Downstream of the Shock (p. 255)).

Graphics and Animations → $\stackrel{\frown}{=}$ Vectors → Set Up...

- 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. 255).

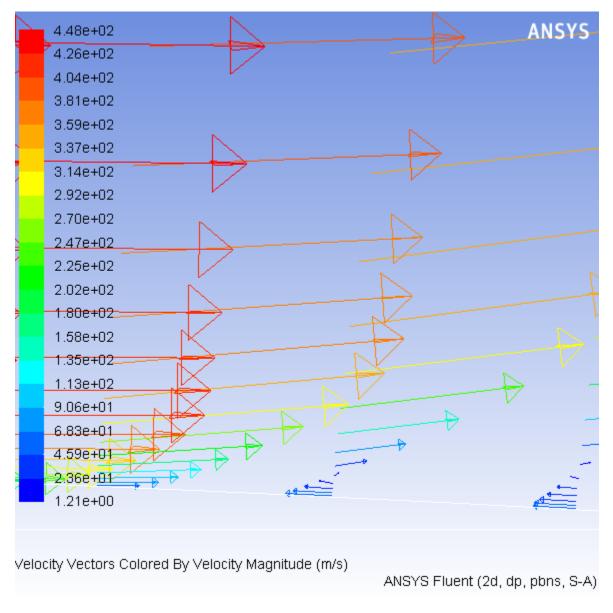


Figure 5.16: Plot of Velocity Vectors Downstream of the Shock

Flow reversal is clearly visible in Figure 5.16: Plot of Velocity Vectors Downstream of the Shock (p. 255).

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 monitors, 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. 123).

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:

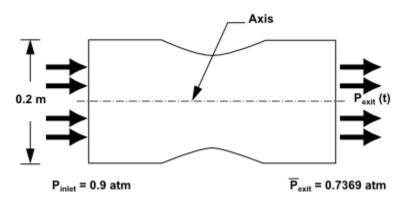
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

and that you are familiar with the ANSYS Fluent navigation pane and menu 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. 258). 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. Due to symmetry, only half of the nozzle is 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. Specifying Solver and Analysis Type
- 6.4.4. Specifying the Models
- 6.4.5. Editing the Material Properties
- 6.4.6. Setting the Operating Conditions
- 6.4.7. Creating the Boundary Conditions
- 6.4.8. Setting the Solution Parameters for Steady Flow and Solving
- 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip the unsteady_compressible_R150 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 Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the **Serial**
- 11. Disable the **Double Precision** option.

6.4.2. Reading and Checking the Mesh

1. Read the mesh file nozzle.msh.

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

The mesh for the half of the geometry is displayed in the graphics window.

2. Check the mesh.

$\mathbf{O}_{\mathsf{General}} \rightarrow \mathsf{Check}$

ANSYS Fluent will perform various checks on the mesh and will report the progress in the console window. Ensure that the reported minimum volume is a positive number.

3. Verify that the mesh size is correct.

 \bigcirc General \rightarrow Scale...

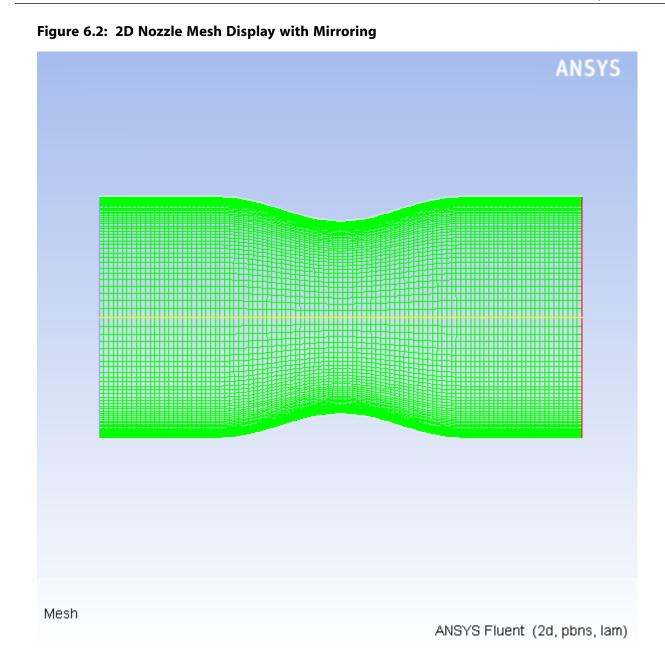
Scale Mesh	—
Domain Extents	Scaling
Xmin (m) -0.1 Xmax (m) 0.3 Ymin (m) 0 Ymax (m) 0.1	 Convert Units Specify Scaling Factors Mesh Was Created In
View Length Unit In	<select> Scaling Factors X 1 Y 1 Scale Unscale</select>
Close Help	

- a. Close the **Scale Mesh** dialog box.
- 4. Mirror the mesh across the centerline (Figure 6.2: 2D Nozzle Mesh Display with Mirroring (p. 261)).

$\clubsuit Graphics and Animations \rightarrow Views...$

Views		—
Views back front Save Name view-0	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes 🖹 🗐 🗐
Apply Camer	ra 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. Specifying Solver and Analysis Type

1. Select the solver settings.



General

Mesh	
Scale	Check Report Quality
Display	
Solver	
Туре	Velocity Formulation
Pressure-Based	Absolute
Oensity-Based	Relative
_	
Time	2D Space
Steady	Planar
Transient	Axisymmetric
	Axisymmetric Swirl
Gravity	Units
Help	

a. Select **Density-Based** from the **Type** list in the **Solver** group box.

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 pressure-based solver is preferred. Also, for transient cases with traveling shocks, the density-based explicit solver with explicit time stepping may be the most efficient.

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.

\bigcirc General \rightarrow Units...

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.

Set Units			<u></u>
Quantities		Units	Set All to
molec-wt moment number-density particles-conc particles-rate percentage power	•	pascal atm psi torr lb/ft2 inches-water	← default si british cgs
pressure pressure-gradient resistance site-density soot-formation-constant-unit	Ŧ	Factor 101325 Offset 0	
New	st	Close Help	

a. Select **pressure** in the **Quantities** selection list.

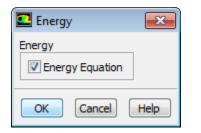
Scroll down the list to find **pressure**.

- b. Select **atm** in the **Units** selection list.
- c. Close the **Set Units** dialog box.

6.4.4. Specifying the Models

1. Enable the energy equation.

$$\mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Energy} \to \mathsf{Edit...}$$



2. Select the k-omega SST turbulence model.

♦ Models → Edit...

Viscous Model		×
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 (5 eqn) Scale-Adaptive Simulation (SAS) k-omega Model Standard SST	Model Constants Alpha*_inf 1 Alpha_inf 0.52 Beta*_inf 0.09 a1 0.31	×
k-omega Options Low-Re Corrections Options Viscous Heating Curvature Correction Production Kato-Launder Production Limiter Intermittency Transition Model	User-Defined Functions Turbulent Viscosity none Prandtl Numbers Energy Prandtl Number None Wall Prandtl Number none Wall Prandtl Number none	
OK	Cancel Help	

- a. Select k-omega (2eqn) in the Model list.
- b. Select **SST** in the **k-omega Model** group box.
- c. Click **OK** to close the **Viscous Model** dialog box.

6.4.5. Editing the Material Properties

1. Set the properties for air, the default fluid material.

 $\clubsuit Materials \rightarrow \overleftarrow{E} air \rightarrow Create/Edit...$

me	Material Type	Order Materials by
air	fluid	Name Chemical Formula
hemical Formula	FLUENT Fluid Materials	Chemical Pormula
	air	 FLUENT Database
	Mixture	User-Defined Database
	none	
operties		_
Density (kg/m3)	ideal-gas 🗸 Edit	Â
Co. (CoostRollingt) (the h)		
Cp (Specific Heat) (j/kg-k)	constant • Edit	
	1006.43	E
Thermal Conductivity (w/m-k)	constant	
	0.0242	
Viscosity (kg/m-s)	constant	
	1.789 4e -05	
		-

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. Setting the Operating Conditions

1. Set the operating pressure.

Generating Conditions → Operating Conditions...

Operating Conditions	×
Pressure	Gravity
Operating Pressure (atm)	Cravity
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. Creating the Boundary Conditions

1. Set the boundary conditions for the nozzle inlet (inlet).

```
\clubsuit Boundary \ Conditions \rightarrow \overleftarrow{\sqsubseteq} inlet \rightarrow Edit...
```

Pressure Inlet		×
Zone Name		
inlet		
Momentum Thermal Radiation Species	DPM Multiphase	uds
Reference Frame At	bsolute	•
Gauge Total Pressure (atm)	.9	constant 🔻
Supersonic/Initial Gauge Pressure (atm)	.7369	constant 👻
Direction Specification Method No	ormal to Boundary	•
Turbulence		
Specification Method Inte	ensity and Viscosity Ratio	▼
	Turbulent Intensity (%	1.5 P
	Turbulent Viscosity Ratio	P 10
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. Set the boundary conditions for the nozzle exit (outlet).



Pressure Outlet
Zone Name
outlet
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (atm) 0.7369 constant
Backflow Direction Specification Method Normal to Boundary
Average Pressure Specification
Target Mass Flow Rate
Non-Reflecting Boundary
Turbulence
Specification Method Intensity and Viscosity Ratio
Backflow Turbulent Intensity (%) 1.5
Backflow Turbulent Viscosity Ratio 10
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. Setting the Solution Parameters for Steady Flow and Solving

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

Solution Methods	
Formulation	
Implicit 🗸	
Flux Type	
Roe-FDS 🔹	
Spatial Discretization	
Gradient	*
Least Squares Cell Based 🔹	
Flow	
Second Order Upwind 🗸	
Turbulent Kinetic Energy	
Second Order Upwind 👻	
Specific Dissipation Rate	
Second Order Upwind 👻	
	Ŧ
Transient Formulation	
· · · · · · · · · · · · · · · · · · ·	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
High Order Term Relaxation Options	
Convergence Acceleration For Stretched Meshes	
Default	
Help	

- 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 drop-down lists.

Second-order discretization provides optimum accuracy.

2. Modify the Courant Number.

Solution Controls

Solution Controls	
Courant Number	
Under-Relaxation Factors	
Turbulent Kinetic Energy	^
0.8	
Specific Dissipation Rate	
0.8	
Turbulent Viscosity	
1	
Solid	
1	
	Ŧ
Default	
Equations Limits Advanced	
Help	

a. Set the **Courant Number** to 50.

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.

 $\clubsuit Monitors \rightarrow \overleftarrow{\sqsubseteq} Residuals \rightarrow Edit...$

Residual Monitors				—
Options Print to Console Plot Window 1 Curves Axes Iterations to Plot 1000	Equations Residual continuity x-velocity y-velocity energy	Monitor V V V	E	
Iterations to Store	Residual Values Normalize Scale Compute Local	Scale	Iterations 5 V	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. Enable the plotting of mass flow rate at the flow exit.

 $\clubsuit Monitors (Surface Monitors) \rightarrow Create...$

Surface Monitor	
Name	Report Type
surf-mon-1	Mass Flow Rate 🔹
Options	Field Variable
Print to Console	Pressure
V Plot	Static Pressure 💌
Window	Surfaces 🔋 🗏 🖃
2 Curves Axes	default-interior
V Write	inlet lower-wall
File Name	outlet
noz_ss.out	symmetry
X Axis	
Iteration 🔻	
Get Data Every	
1 Iteration •	
Average Over	
1	New Surface 💌
OK Cancel Help	

a. Enable **Plot** and **Write**.

Note

When **Write** is enabled in the **Surface Monitor** dialog box, the mass flow rate history will be written to a file. If you do not enable the write option, the history information will be lost when you exit ANSYS Fluent.

- b. Enter noz_ss.out for File Name.
- c. Select Mass Flow Rate in the Report Type drop-down list.
- d. Select outlet in the Surfaces selection list.
- e. Click **OK** to close the **Surface Monitor** dialog box.
- 5. Save the case file (noz_ss.cas.gz).

File \rightarrow Write \rightarrow Case...

6. Initialize the solution.

Solution Initialization

Solution Initialization
Initialization Methods
 Hybrid Initialization Standard Initialization
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.
- 7. Set up gradient adaption for dynamic mesh refinement.

Adapt → Gradient...

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.

Gradient Adaption				x
Options	Method Curvature Gradient Iso-Value Normalization Standard Scale Normalize	Gradients of Pressure Static Pressure Min 0 Coarsen Threshold 0.3	Max 0 Refine Threshold 0.7	• •
Adap	Dynamic Dynamic Interval 100 T 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 readjustment 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 Coarsen	Zones	 Min Cell Volume (m3) 0 Min # of Cells 0 ✓ Max # of Cells 20000 ✓ Max Level of Refine 2 ✓ Volume Weight 1
	OK Cancel H	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.

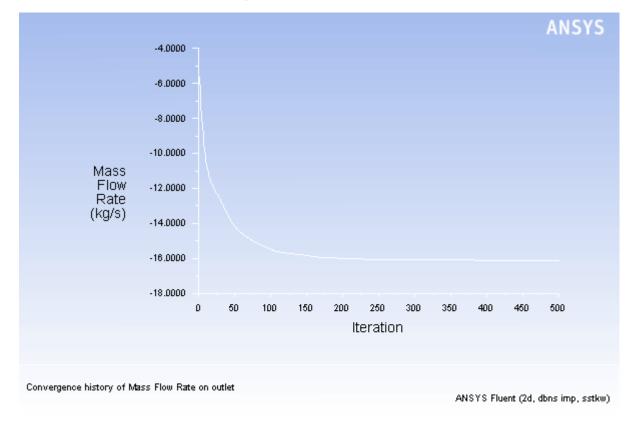
iii. Click **OK** to save your settings and close the **Mesh Adaption Controls** dialog box.

- j. Click Close to close the Gradient Adaption dialog box.
- 8. Start the calculation by requesting 500 iterations.

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.

Figure 6.3: Mass Flow Rate History



9. Save the case and data files (noz_ss.cas.gz and noz_ss.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

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.

Graphics and Animations → \blacksquare Mesh → Set Up...

The Mesh Display dialog box appears.

- 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 graphic windows (Figure 6.4: 2D Nozzle Mesh after Adaption (p. 277))

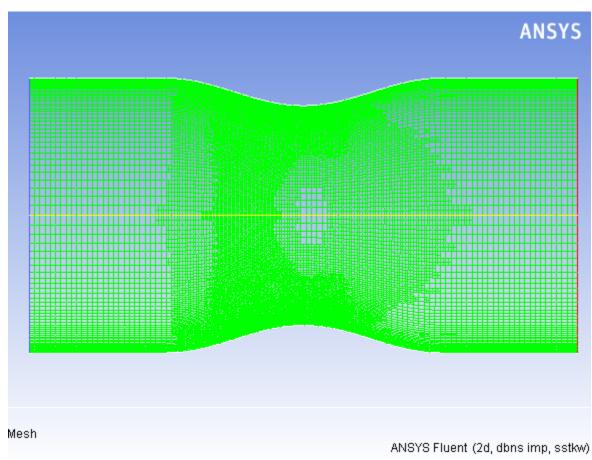


Figure 6.4: 2D Nozzle Mesh after Adaption

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. 279)).

• Graphics and Animations $\rightarrow \overline{\Xi}$ Contours \rightarrow Set Up...

Contours		x	
Options	Contours of		
V Filled	Pressure	•	
Vode Values Global Range	Static Pressure	•	
Auto Range	Min Max		
Clip to Range	0		
Draw Mesh	Surfaces 🔋 🗎		
	default-interior	-	
Levels Setup	inlet		
20 1	ower-wall		
	symmetry		
Surface Name Pattern	New Surface		
Ma	Surface Types		
	axis		
	clip-surf	1	
	exhaust-fan	1	
	fan 🗸 🗸		
Display Compute Close Help			

- a. Enable **Filled** in the **Options** group box.
- b. Click **Display** and close the **Contours** dialog box.

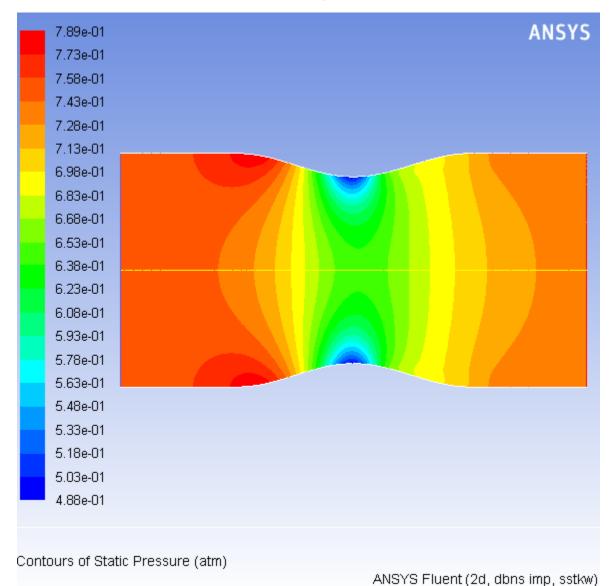


Figure 6.5: Contours of Static Pressure (Steady Flow)

The steady flow prediction in Figure 6.5: Contours of Static Pressure (Steady Flow) (p. 279) shows the ex-

13. Display the steady-flow velocity vectors (Figure 6.6: Velocity Vectors (Steady Flow) (p. 281)).

Graphics and Animations → $\boxed{=}$ Vectors → Set Up...

pected pressure distribution, with low pressure near the nozzle throat.

Vectors		x
Options	Vectors of	
🗸 Global Range	Velocity	•
V Auto Range	Color by	
Clip to Range	Velocity	•
Draw Mesh	Velocity Magnitude	•
Style	Min Max	
arrow 🔻	0	
Scale Skip	Surfaces	
1 0 🔺	default-interior	
Vector Options	inlet	
Vector Options	lower-wall	
Custom Vectors	outlet symmetry	
	Synnicuty	
Surface Name Pattern		
Match	New Surface 🔻	
	Surface Types	
	axis	
	clip-surf	
	exhaust-fan	
	fan	*
Display	Compute Close Help	

- a. Retain all default settings.
- b. Click **Display** and close the **Vectors** dialog box.

You can zoom in to view the recirculation of the velocity vectors.

The steady flow prediction in Figure 6.6: Velocity Vectors (Steady Flow) (p. 281) shows the expected form, with a peak velocity of approximately 300 m/s through the nozzle.

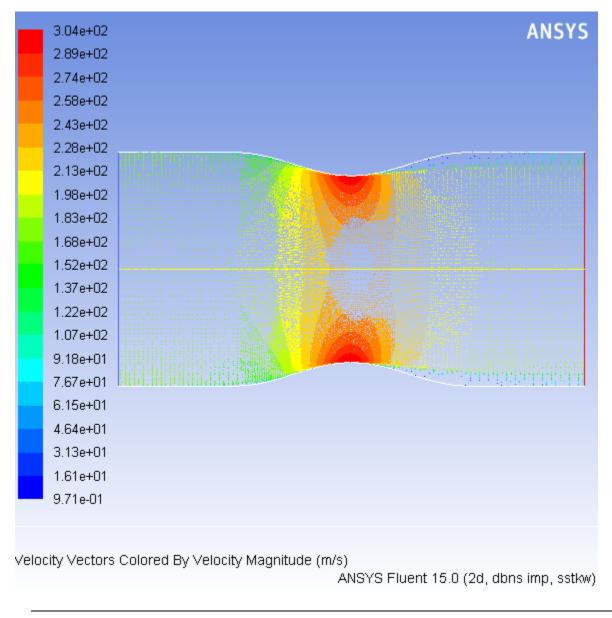


Figure 6.6: Velocity Vectors (Steady Flow)

Note

To improve the clarity of the flow pattern, you can increase the size of the displayed velocity vectors by increasing the value in the **Scale** field.

14. Check the mass flux balance.



Warning

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.

Flux Reports		—	
Options Mass Flow Rate Total Heat Transfer Rate	Boundaries 🗎 🔳 🚍 default-interior inlet	Results	
Radiation Heat Transfer Rate	lower-wall outlet	-16.09498405456543	
Boundary Types	symmetry		
Boundary Name Pattern Match			
Save Output Parameter		Net Results (kg/s) -0.0002403259	
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.

Warning

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.



General	
Mesh Scale Display	Check Report Quality
Type Pressure-Based Opensity-Based	Velocity Formulation Absolute Relative
Time ◎ Steady ◎ Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

a. Select Transient in the Time list.

2. Read the user-defined function (pexit.c), in preparation for defining the transient condition for the nozzle exit.

Define \rightarrow **User-Defined** \rightarrow **Functions** \rightarrow **Interpreted...**

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

ω	=	circular frequency of transient pressure (rad/s)
\overline{p}_{exit}	=	mean exit pressure (atm)

In this case, $\omega = 2200$ rad/s, and $\overline{p}_{exit} = 0.7369$ atm.

A user-defined function (pexit.c) has been written to define the equation (Equation 6.1 (p. 283)) required for the pressure profile.

Note

To input the value of Equation 6.1 (p. 283) 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 UDF Manual.

Interpreted UDFs	×
Source File Name	
pexit.c	Browse
CPP Command Name	
срр	
Stack Size	
Display Assembly Listing	
Use Contributed CPP	
Interpret Close	Help

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. Set the transient boundary conditions at the nozzle exit (outlet).

$\textcircled{Boundary Conditions} \rightarrow \fbox{outlet} \rightarrow \texttt{Edit...}$

Pressure Outlet	×
Zone Name	
outlet	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Gauge Pressure (atm) udf transient_pressure	•
Backflow Direction Specification Method Normal to Boundary	•
Target Mass Flow Rate	_
Non-Reflecting Boundary	
Turbulence	
Specification Method Intensity and Viscosity Ratio	•
Backflow Turbulent Intensity (%) 1.5	e
Backflow Turbulent Viscosity Ratio	e
OK Cancel 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.

Adapt → 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).

- 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 plotting of the mass flow rate at the nozzle exit.

♦ Monitors (Surface Monitors) $\rightarrow =$ surf-mon-1 \rightarrow Edit...

Because each time step requires 10 iterations, a smoother plot will be generated by plotting at every time step.

Surface Monitor	
Name	Report Type
surf-mon-1	Mass Flow Rate 💌
Options	Field Variable
Print to Console	Pressure
V Plot	Static Pressure 💌
Window	Surfaces 🔋 🗏 🖃
3 Curves Axes	default-interior
V Write	inlet lower-wall
File Name	outlet
noz_uns.out	symmetry
X Axis	
Time Step	
Get Data Every	
1 Time Step	
Average Over(Time Steps)	
	New Surface 🔻
OK	Cancel Help

- a. Set **Window** to **3**.
- b. Enter noz_uns.out for File Name.
- c. Select Time Step from the X Axis drop-down list.
- d. Select Time Step from the Get Data Every drop-down list.
- e. Click OK to close the Surface Monitor dialog box.
- 2. Save the transient solution case file (noz_uns.cas.gz).

File \rightarrow Write \rightarrow Case...

3. Modify the plotting of residuals.

Monitors → Eresiduals → Edit...

- 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.
- 4. Set the time step parameters.

Run Calculation

The selection of the time step is critical for accurate time-dependent flow predictions. Using a time step of 2.85596x10⁻⁵ seconds, 100 time steps are required for one pressure cycle. The pressure cycle begins and ends with the initial pressure at the nozzle exit.

Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method Fixed •	Time Step Size (s) 2.85596e-5
Settings	Number of Time Steps
Options	
Extrapolate Variables Data Sampling for Time Sampling Interval	Sampling Options
Max Iterations/Time Step	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- 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.

Warning

Calculating 600 time steps will require significant CPU resources. Instead of calculating the solution, you can read the data file (noz_uns.dat.gz) with the precalculated solution. This data file can be found in the folder where you found the mesh and UDF files.

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. 288).

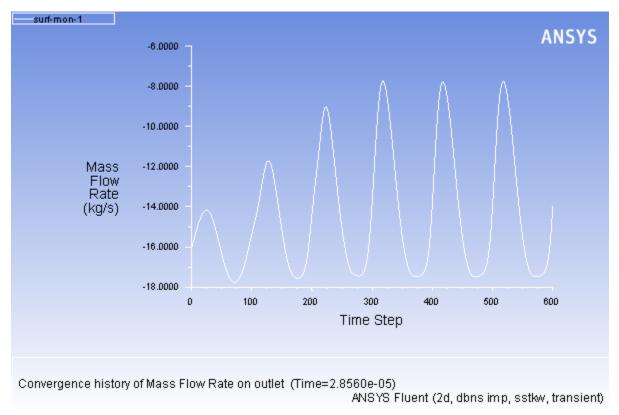


Figure 6.7: Mass Flow Rate History (Transient Flow)

- 5. Optionally, you can review the effect of dynamic mesh adaption performed during transient flow computation as you did in steady-state flow case.
- 6. Save the transient case and data files (noz_uns.cas.gz and noz_uns.dat.gz).

File \rightarrow Write \rightarrow 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.

\bigcirc Calculation Activities (Autosave Every) \rightarrow Edit...

🖬 Autosave 💽
Save Data File Every (Time Steps) 10
Data File Quantities
Save Associated Case Files
 Only if Modified Each Time
File Storage Options
Retain Only the Most Recent Files
Maximum Number of Data Files 🚺 🗨
Only Associated Case Files are Retained
File Name
noz_anim Browse
Append File Name with time-step
OK Cancel Help

- 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 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. This will yield file names of the form noz_anim-1-00640.cas and noz_anim-1-00640.dat, where 00640 is the time step number.

Optionally, you can add the extension .gz to the end of the file name (for example, noz_anim.gz), which will instruct ANSYS Fluent to save the case and data files in compressed format, yielding file names of the form noz_anim-1-00640.cas.gz.

e. Click **OK** to close the **Autosave** dialog box.

Extra

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 animation sequences for the nozzle pressure and Mach number contour plots.

 \bigcirc Calculation Activities (Solution Animations) \rightarrow Create/Edit...

💶 Soluti	ion Animation				×
Animation	Sequences 2				
Active	Name	Every	When		^
	pressure	1	Time Step	Define	
	mach-number	1	Time Step	Define	
	sequence-3) Iteration	Define	
	sequence-4		Iteration	• Define	
	sequence-5] Iteration	▼ Define	Ŧ
OK Cancel Help					

- a. Set Animation Sequences to 2.
- b. Enter pressure for the Name of the first sequence and mach-number for the second sequence.
- c. Select Time Step from the When drop-down lists for both sequences.

The default value of 1 in the **Every** integer number entry box instructs ANSYS Fluent to update the animation sequence at every time step.

d. Click the **Define...** button for **pressure** to open the associated **Animation Sequence** dialog box.

Animation Sequence	×	
Sequence Parameters	Display Type	
Storage Type In Memory Metafile PPM Image Storage Directory	 Mesh Contours Pathlines Particle Tracks Vectors XY Plot Monitor Monitor Type Residuals Create < Edit 	
OK Cancel Help		

i. Select In Memory from the Storage Type group box.

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.

ii. Enter 4 for **Window** and click the **Set** button.

iii. Select **Contours** from the **Display Type** group box to open the **Contours** dialog box.

Contours	X		
Options	Contours of		
Filled	Pressure 🔻		
 Node Values Global Range 	Static Pressure		
Auto Range	Min Max		
Clip to Range	0.25 1.25		
Draw Mesh	Surfaces		
Levels Setup	default-interior inlet lower-wall outlet		
	symmetry		
Surface Name Pattern Match	New Surface 💌		
Match	Surface Types		
	axis		
	clip-surf		
	exhaust-fan fan +		
	- · ·		
Display	Compute Close Help		

- A. Ensure that **Filled** is enabled in the **Options** group box.
- B. Disable Auto Range.
- C. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** dropdown lists.
- D. Enter 0.25 atm for Min and 1.25 atm for Max.

This will set a fixed range for the contour plot and subsequent animation.

E. Click **Display** and close the **Contours** dialog box.

Figure 6.8: Pressure Contours at t=0.017136 s (p. 292) shows the contours of static pressure in the nozzle after 600 time steps.

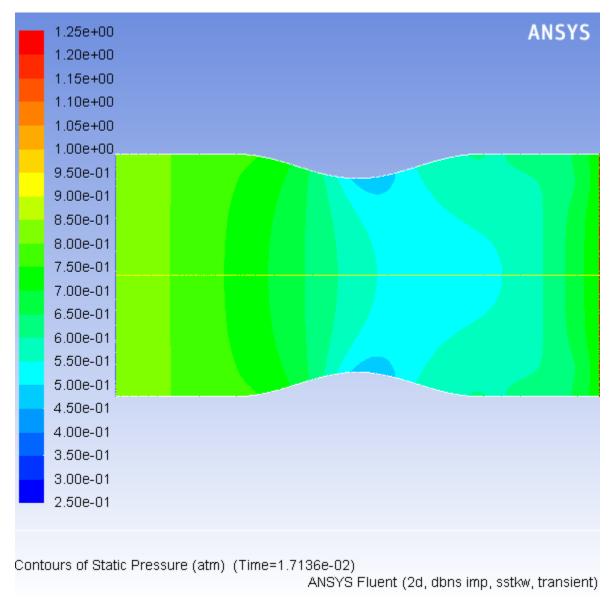


Figure 6.8: Pressure Contours at t=0.017136 s

- iv. Click OK to close the Animation Sequence dialog box associated with the pressure sequence.
- e. Click the **Define...** button for **mach-number** to open the associated **Animation Sequence** dialog box.
 - i. Ensure that In Memory is selected in the Storage Type list.
 - ii. Enter 5 for **Window** and click the **Set** button.
 - iii. Select **Contours** in the **Display Type** group box to open the **Contours** dialog box.
 - A. Select Velocity... and Mach Number from the Contours of drop-down lists.
 - B. Ensure that **Filled** is enabled from the **Options** group box.
 - C. Disable Auto Range.

- D. Enter 0.00 for **Min** and 1.30 for **Max**.
- E. Click **Display** and close the **Contours** dialog box.

Figure 6.9: Mach Number Contours at t=0.017136 s (p. 293) shows the Mach number contours in the nozzle after 600 time steps.

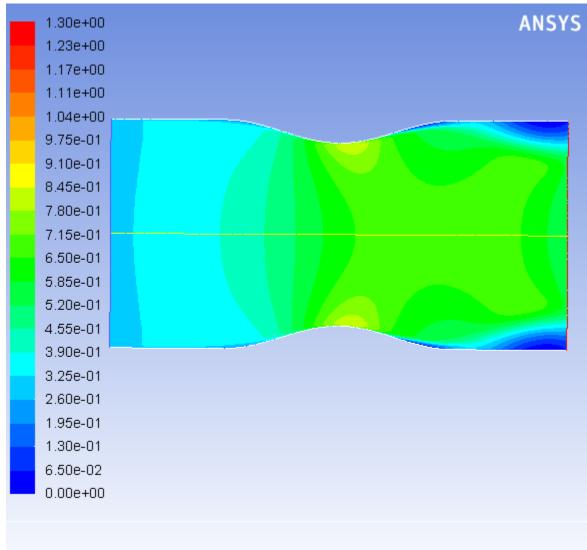


Figure 6.9: Mach Number Contours at t=0.017136 s

Contours of Mach Number (Time=1.7136e-02) ANSYS Fluent (2d, dbns imp, sstkw, transient)

iv. Click OK to close the Animation Sequence dialog box associated with the mach-number sequence.

- f. Click **OK** to close the **Solution Animation** dialog box.
- 3. Continue the calculation by requesting 100 time steps.

Run Calculation

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
Options	
Extrapolate Variables Data Sampling for Time Sampling Interval Time Sampled (s	Sampling Options
Max Iterations/Time Step	Reporting Interval
Profile Update Interval	
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.

4. Change the display options to include double buffering.

\bigcirc Graphics and Animations \rightarrow Options...

Double buffering will allow for a smoother transition between the frames of the animations.

C Display Options	
Rendering	Graphics Window
Line Width 1 Point Symbol (+)	Active Window Open 4 Set
Animation Option Wireframe	Color Scheme Workbench 👻
 Double Buffering Outer Face Culling Hidden Line Removal Hidden Surface Removal Removal Method Hardware Z-buffer Display Timeout Timeout in seconds 	Lighting Attributes
Apply Info Lights	Close Help

- a. Retain the **Double Buffering** option in the **Rendering group** box.
- b. Enter 4 for **Active Window** and click the **Set** button.

Note

Alternatively, you can change the active window using the drop-down list at the top of the graphics window.

- c. Click Apply and close the Display Options dialog box.
- 5. Play the animation of the pressure contours.

Graphics and Animations → $\overline{\equiv}$ Solution Animation Playback → Set Up...

Playback	—
Playback	Animation Sequences
Playback Mode Play Once	Sequences pressure mach-number
I ← Frame	
< •	
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 window 4 is visible in the viewer. If it is not, select it from the drop-down list at the top left of the viewer 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. 297) and Figure 6.11: Pressure Contours at t=0.019135 s (p. 298).

6. In a similar manner to steps 4 and 5, select the appropriate active window and 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. 299) and Figure 6.13: Mach Number Contours at t=0.019135 s (p. 300)...

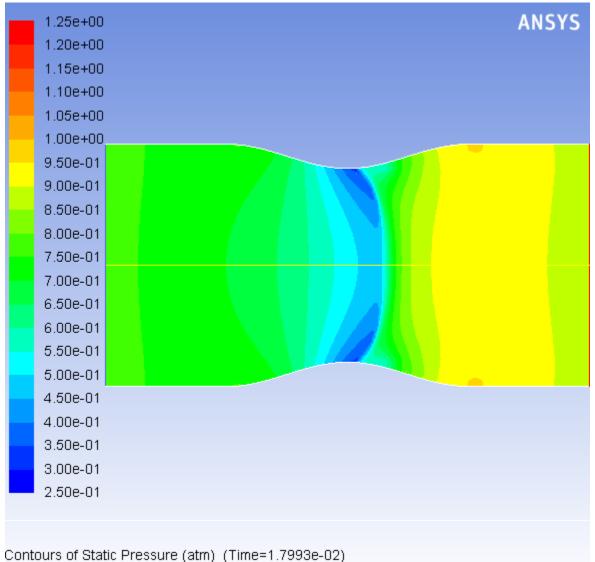
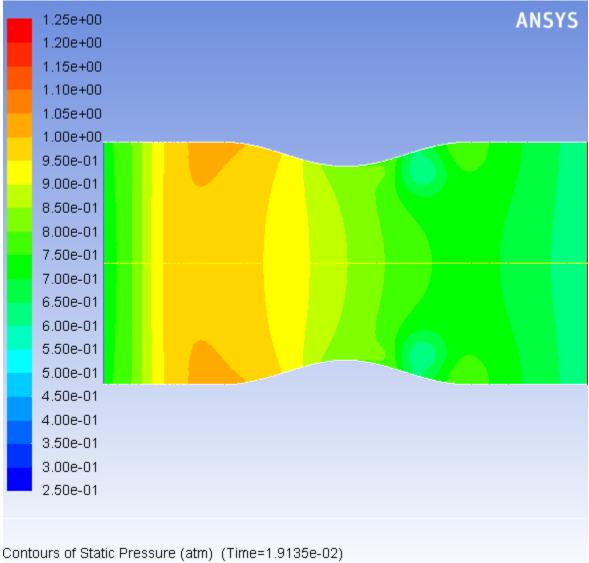


Figure 6.10: Pressure Contours at t=0.017993 s

ANSYS Fluent (2d, dbns imp, sstkw, transient)





ANSYS Fluent (2d, dbns imp, sstkw, transient)

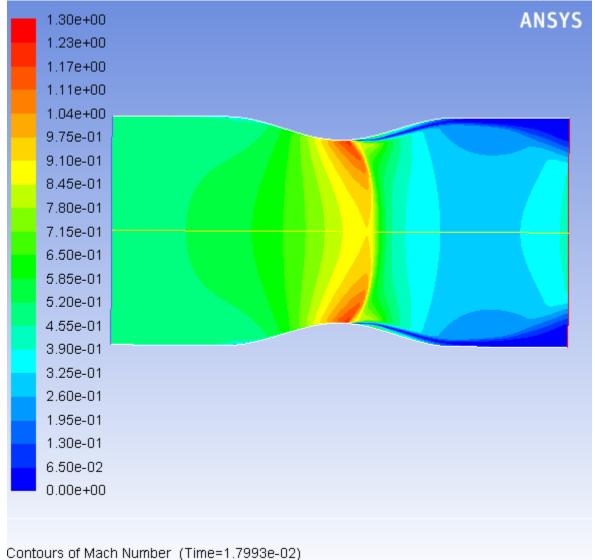


Figure 6.12: Mach Number Contours at t=0.017993 s

ANSYS Fluent (2d, dbns imp, sstkw, transient)

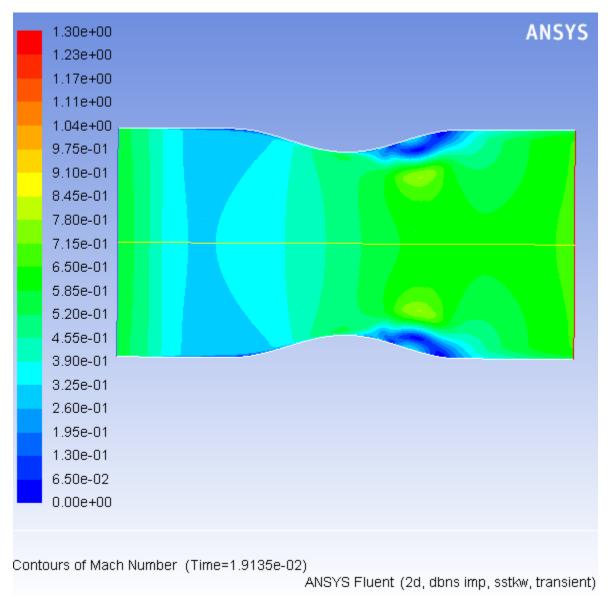


Figure 6.13: Mach Number Contours at t=0.019135 s

Extra

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

Since 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_anim-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. 302)).

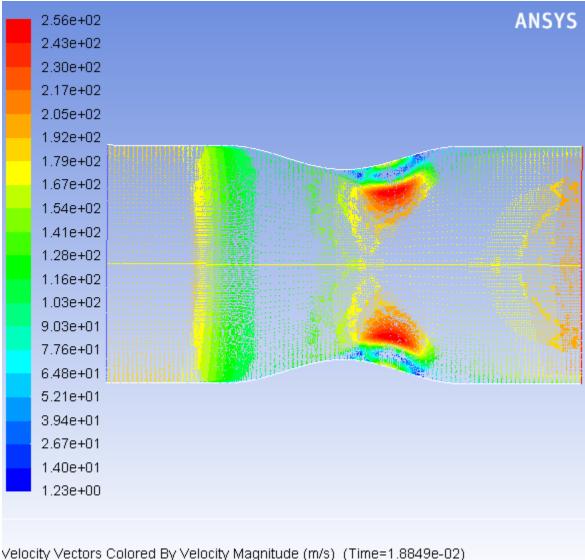
Vectors		—	
Options	Vectors of		
🔽 Global Range	Velocity	-	
Auto Range	Color by		
Clip to Range	Velocity		
Draw Mesh	Velocity Magnitude	•	
Style	Min (m/s)	Max (m/s)	
arrow	1.234463	255.6297	
Scale Skip	Surfaces		
1 0 🛋	default-interior		
Vector Options	inlet		
	lower-wall outlet		
Custom Vectors	symmetry		
Surface Name Pattern			
Match	New Surface		
	Surface Types		
	axis	<u> </u>	
	clip-surf exhaust-fan		
	fan	-	
	I - .		
Display	Compute Close	Help	

Graphics and Animations → \blacksquare Vectors → Set Up...

- a. Ensure Auto Scale is enabled under Options.
- b. Retain the default values for all other properties.

c. Click **Display** and close the **Vectors** dialog box.





ANSYS Fluent (2d, dbns imp, sstkw, transient)

The transient flow prediction in Figure 6.14: Velocity Vectors at t=0.018849 s(p. 302) shows the expected form, with peak velocity of approximately 241 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. You learned how to generate a steady-state solution as an initial condition for the transient case, and how to set solution parameters for implicit time-stepping.

You also learned how to manage the file saving and graphical postprocessing for time-dependent flows, using file autosaving to automatically save solution information as the transient calculation proceeds.

Finally, you learned how to use ANSYS Fluent's solution animation tool to create animations of transient data, and how to view the animations using the playback feature.

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 independent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123).

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. 123)

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

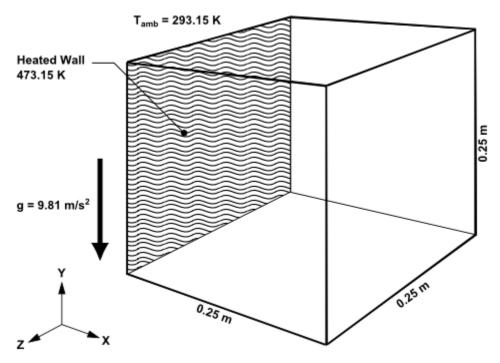
7.3. Problem Description

The problem to be considered is shown schematically in Figure 7.1: Schematic of the Problem (p. 306). A three-dimensional box $(0.25 \text{ m} \times 0.25 \text{ m} \times 0.25 \text{ m})$ 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 1×10^8 . 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. Specifying Solver and Analysis Type
- 7.4.3. Specifying Solver and Analysis 1 7.4.4. Specifying the Models
- 7.4.5. Defining the Materials
- 7.4.6. Specifying Boundary Conditions
- 7.4.7. Obtaining the Solution
- 7.4.8. Postprocessing
- 7.4.9. Comparing the Contour Plots after Varying Radiating Surfaces
- 7.4.10. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip radiation_natural_convection_R150.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** 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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Run in single precision (disable Double Precision).
- 11. Ensure that Serial is selected under Processing Options.

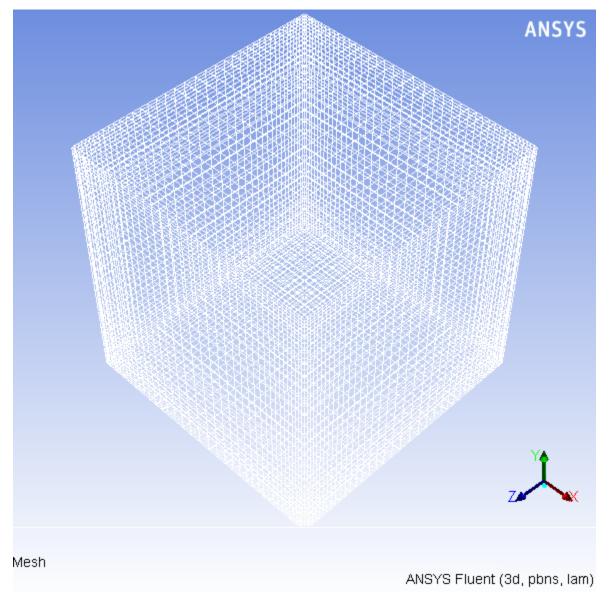
7.4.2. Reading and Checking the Mesh

1. Read the mesh file rad.msh.gz.

$\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

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.

Figure 7.2: Graphics Display of Mesh



2. Check the mesh.

$\clubsuit General \rightarrow Check$

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. Specifying Solver and Analysis Type

1. Confirm the solver settings and enable gravity.



General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Type Pressure-Based Density-Based	0
Time	
🔽 Gravity	Units
Gravitational Acceler	
X (m/s2)	P
Y (m/s2) -9.81	e
Z (m/s2) 0	e
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 m/s^2 for **Y** in the **Gravitational Acceleration** group box.

7.4.4. Specifying the Models

1. Enable the energy equation.

 $\textcircled{Models} \rightarrow \overleftarrow{E} Energy \rightarrow Edit...$

💶 Energy	_X
Energy	
Energy Equation	
OK Cancel	Help

2. Set up the **Surface to Surface (S2S)** radiation model.

```
 \mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Radiation} \to \mathsf{Edit...}
```

a. Select **Surface to Surface (S2S)** from the **Model** list.

You will be prompted with a message box directing you to click **OK** in the **Radiation Model** dialog box and re-open it to set the S2S options. When you re-open the dialog box, the additional inputs for the S2S model will be visible.

Radiation Model	×
Model	Iteration Parameters
 ○ Off ○ Rosseland 	Energy Iterations per Radiation Iteration 10
 P1 Discrete Transfer (DTRM) 	Maximum Number of Radiation Iterations 5
 Surface to Surface (S2S) Discrete Ordinates (DO) 	Residual Convergence Criteria 0.001
	View Factors and Clustering
	Settings
	Compute/Write/Read
	Read Existing File
Solar Load	
Model	
Off	
Solar Ray Tracing	
O DO Irradiation	
Solar Calculator	
	OK Cancel Help

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.

b. 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 Clustering		
Clustering		
Options		
 Manual Automatic 		
Manual	Automatic	
Faces per Surface Clust for Flow Boundary Zone 1 Apply to All Walls		
View Factors		
Basis	Method	
 Face to Face Cluster to Cluster 	 Ray Tracing Hemicube 	
Surfaces	Parameters	
 Blocking Nonblocking 	Resolution 10	
	Subdivisions 5	
	Normalized Separation 5 Distance	
Zones Participating in View Factor Calculation	Select	
ОК	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.
- c. 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.

d. Click **OK** to close the **Radiation Model** dialog box.

7.4.5. Defining the Materials

1. Set the properties for air.

	≣ air →	Create/Edit
--	----------------	-------------

Create/Edit Materials				— ×
Name air Chemical Formula	Flue air	ent Fluid Materials ture		Order Materials by Name Chemical Formula Fluent Database User-Defined Database
Properties				
Cp (Specific Heat) (j/kg-k)	constant 1021	•	Edit	
Thermal Conductivity (w/m-k)	constant	•	Edit	
Viscosity (kg/m-s)	constant	•	Edit	
Molecular Weight (kg/kgmol)	2.485e-05 constant	•	Edit	
	28.966		•	
	Change/Create	Delete Close	Help	

- a. Select incompressible-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**.

 $\textcircled{} Materials \rightarrow \fbox{} Solid \rightarrow Create/Edit...$

me	Material Trans	Order Materials by
nsulation	Solid Solid	 Name
emical Formula	Fluent Solid Materials	Chemical Formula
	insulation	Fluent Database
	Mixture	User-Defined Database
operties	none	*
Density (kg/m3) constant 50 Cp (Specific Heat) (j/kg-k) constant 800	Edit	
Thermal Conductivity (w/m-k) constant 0.09	Edit	

- a. Enter insulation for Name.
- b. Delete the entry in the **Chemical Formula** field.
- c. Enter 50 kg/m³ 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. Specifying Boundary Conditions

1. Set the boundary conditions for the front wall (w-high-x).

$\clubsuit Boundary \ Conditions \rightarrow \fbox w-high-x \rightarrow Edit...$

Boundary Con	ditions	
Zone		
default-interior w-high-x w-high-y w-high-z w-low-x w-low-y w-low-y w-low-z		
Phase mixture Edit Parameters Display Mesh Highlight Zone	Type wall Copy Profiles Operating Conditions Periodic Conditions	ID 17
Help		

The Wall dialog box appears.

💶 Wall			
Zone Name			
w-high-x			
Adjacent Cell Zone			
fluid			
Momentum Thermal	Radiation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
Heat Flux	Heat Transfer Coefficient (w	v/m2-k) 5	constant 👻
 Temperature Convection 	Free Stream Temperat	ture (k) 293.15	constant 💌
 Radiation Mixed 	External Em	issivity 0.75	constant 🔹
i via System Coupling	9		
Material Name	External Radiation Temperat	ture (k) 293.15	constant
insulation	▼ Edit Internal Em	issivity 0.95	constant 👻
		Wall	Thickness (m) 0.05
	Heat Generation Rate	(w/m3)	
		0	constant •
			Shell Conduction Define
	ОК С	ancel Help	
	UK C	nep	

- 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/m²-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.

Generations → Copy...

Copy Condition: From Boundary Zone w-high-x w-high-y w-high-z w-low-x w-low-y w-low-y w-low-z	s To Boundary Zones 🗐 🚍 w-high-y w-high-z w-low-x w-low-y w-low-z
Сору	Close Help

- a. Select w-high-x from 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.
- e. Close the Copy Conditions dialog box.
- 3. Set the boundary conditions for the heated wall (**w-low-x**).

💶 Wall				2
Zone Name				
w-low-x				
Adjacent Cell Zone				
fluid				
Momentum Thermal Rad	iation Species DPM Multiphase U	JDS Wall Film		
Thermal Conditions				
Heat Flux	Temperature (k)	473.15	constant	-
Temperature Convection	Internal Emissivity	0.95	constant	-
 Radiation Mixed via System Coupling 		Wall Thickness ((m) 0	F
Material Name	Heat Generation Rate (w/m3)	0	constant	-
aluminum	▼ Edit		Shell Conduction	Define
	OK	Help		

$\textcircled{} Boundary \ Conditions \rightarrow \overleftarrow{\sqsubseteq} w-low-x \rightarrow Edit...$

- 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**).

$\mathbf{\hat{\mathbf{V}}}$ Boundary Conditions $\rightarrow \mathbf{E}$ w-high-y \rightarrow Edi

💶 Wall			
Zone Name			
w-high-y			
Adjacent Cell Zone			
fluid			
the second la	to be a femiliant for	a lucana l	
Momentum Thermal Rad	diation Species DPM Multiphase UD	DS Wall Film	1
Thermal Conditions			
Heat Flux	Heat Transfer Coefficient (w/m2	2-k) 3	constant 💌
 Temperature Convection 	Free Stream Temperature	(k) 293.15	constant 🔻
Radiation			constant
Mixed	External Emissiv	vity 0.75	constant 🔹
via System Coupling	External Radiation Temperature	(k) 293.15	
Material Name		293.15	constant
aluminum	Edit Internal Emissiv	vity 0.95	constant 💌
		Wall Thickness	(m)
			0.05 P
	Heat Generation Rate (w/n	m3) 0	constant 🔻
			Shell Conduction Define
			Define
	OK	Help	

- 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/m²-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.

€Boundary	Conditions	\rightarrow	Copy	v
• Doundary	Conditions		COP	y

Copy Condition	s 💌
From Boundary Zone w-high-x w-high-y w-high-z w-low-x w-low-y w-low-y	To Boundary Zones 🗐 🚍 w-high-x w-high-z w-low-x w-low-y w-low-z
Сору	Close Help

- a. Select w-high-y from 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.

7.4.7. Obtaining the Solution

1. Set the solution parameters.

```
Solution Methods
```

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	-
Least Squares Cell Based 🔹	
Pressure	
Body Force Weighted 🔹	
Momentum	Ξ
Second Order Upwind 🔻	
Energy	
Second Order Upwind	
	Ŧ
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
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. Initialize the solution.

Solution Initialization

Solution Initialization
Initialization Methods
 Hybrid Initialization Standard Initialization
More Settings Initialize
Patch
Reset DPM Sources Reset Statistics
Help

a. Retain the default selection of Hybrid Initialization from the Initialization Methods list.

b. Click Initialize.

3. Define a surface monitor to aid in judging convergence.

It is good practice to use monitors of physical solution quantities together with residual monitors when determining whether a solution is converged. In this step you will set up a surface monitor of the average temperature on the z=0 plane.

a. Create the new surface, **zz_center_z**.

Surface \rightarrow	Iso-Surface
-----------------------	-------------

💶 Iso-Surface			- ×-
Surface of Constant Mesh Z-Coordinate Min (m) Max (m) -0.125 0.125	• •	From Surface default-interior w-high-x w-high-y w-high-z w-low-x w-low-y	
Iso-Values (m) 0 New Surface Name zz_center_z	F	From Zones	
Create Compute N	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 monitor.

.

Gerea (Surface Monitors) → Crea	to
---------------------------------	----

💶 Surface Monitor	—		
Name	Report Type		
surf-mon-1	Area-Weighted Average		
Options	Field Variable		
Print to Console	Temperature 🔻		
V Plot	Static Temperature		
Window	Surfaces		
2 Curves Axes	default-interior w-high-x		
C Write	w-high-y		
File Name	w-high-z		
surf_mon_1.out	w-low-x		
N A.S.	w-low-y w-low-z		
X Axis Iteration	zz_center_z		
Get Data Every			
1 Iteration V			
Average Over	Highlight Surfaces		
1	New Surface		
ОК	Cancel Help		

- i. Retain the default entry of **surf-mon-1** for the **Name** of the surface monitor.
- ii. Enable the **Plot** option.

Note

Unlike residual values, data from other monitors is not saved as part of the solution set when the ANSYS Fluent data file is saved. If you want to access the surface monitor data in future ANSYS Fluent sessions, you can enable the **Write** option and enter a **File Name** for the monitor output.

- iii. Select Area-Weighted Average from the Report Type drop-down list.
- iv. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- v. Select **zz_center_z** from the **Surfaces** selection list.
- vi. Click **OK** to save the surface monitor settings and close the **Surface Monitor** dialog box.
- 4. Save the case file (rad_a_1.cas.gz)

```
File \rightarrow Write \rightarrow Case...
```

5. Start the calculation by requesting 300 iterations.

Run Calculation

Run Calculation
Check Case Preview Mesh Motion
Pseudo Transient Options
Fluid Time Scale
Time Step Method Pseudo Time Step (s) Image: Optimized state Image: Optimized state Image: Optimized state Image: Optimized state
Number of Iterations Reporting Interval
Profile Update Interval
Data File Quantities Acoustic Signals
Calculate
Help

- a. Select User Specified from the Time Step Method list.
- b. Retain the default value of 1 for **Pseudo Time Step**.
- c. Enter 300 for Number of Iterations.
- d. Click Calculate.

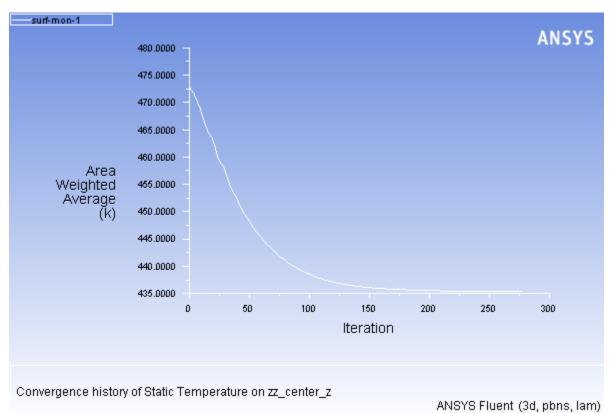
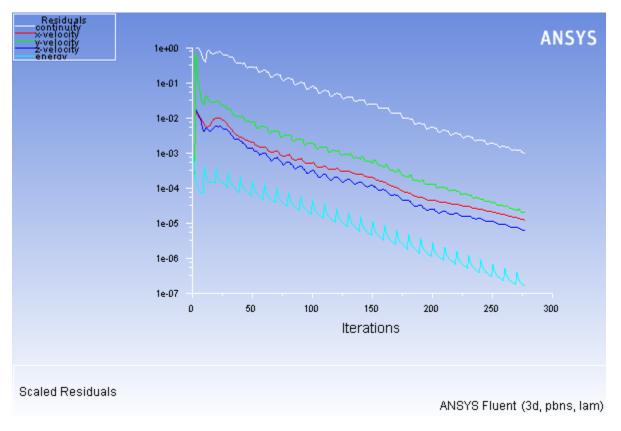


Figure 7.3: Temperature Surface Monitor

The surface monitor 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. 324)) by selecting **Scaled Residuals** from the graphics window drop-down list.

Figure 7.4: Scaled Residuals



7.4.8. Postprocessing

1. Create a new surface, **zz_x_side**, which will be used later to plot wall temperature.

Surface → Line/Rake...

💶 Line/	Rake Surface	— ×-
Options		Number of Points
End Point	ts	
x0 (m)	-0.125	x1 (m) 0.125
y0 (m)	0	y1 (m) 0
z0 (m)	0.125	z1 (m) 0.125
	Select Point	s with Mouse
New Surf	face Name	
zz_x_si	de	
Cre	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.

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.
- 2. Display contours of static temperature.

Contours			- ×
Options Filled Global Range Auto Range Clip to Range Draw Profiles Draw Mesh	Contours of Temperature Static Temperature Min 421 Surfaces w-low-x w-low-y	Max 473.15	
Levels Setup 20 I	w-low-z		E
Surface Name Pattern	New Surface Surface Types axis clip-surf exhaust-fan fan -		
Display Compute Close Help			

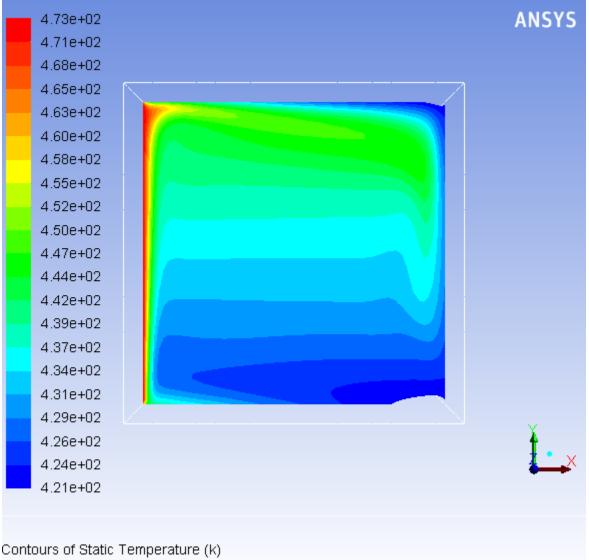
\bigcirc Graphics and Animations → **≡** Contours → Set Up...

- 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. Select **Outline** from the **Edge Type** list.
 - ii. Click **Display** and close the **Mesh Display** dialog box.

The outline of the geometry is displayed in the graphics window.

- e. Disable the Auto Range option.
- f. Enter 421 K for **Min** and 473.15 K for **Max**.
- g. Click **Display** and rotate the view as shown in Figure 7.5: Contours of Static Temperature (p. 326).





ANSYS Fluent (3d, pbns, lam)

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. 326) reveals that the solution appears as expected.

3. Display contours of wall temperature (surfaces in contact with the fluid).



Contours		×
Options	Contours of	
Filled	Temperature	-
 Node Values Global Range 	Wall Temperature	•
Auto Range	Min (k) Max ((k)
Clip to Range	413 473	.15
Draw Profiles Draw Mesh	Surfaces	
	w-low-x	<u></u>
Levels Setup	w-low-y	
	w-low-z zz_center_z	=
	zz_x_side	-
Surface Name Pattern	New Surface	
Match	Surface Types	
	axis	
	clip-surf	
	exhaust-fan fan	
	- ·	•
Display	Compute Close H	Help

- a. Ensure that the **Filled** option is enabled in the **Options** group box.
- b. Disable the Node Values option.
- c. Select Temperature... and Wall Temperature from the Contours of drop-down lists.
- d. Select all surfaces except **default-interior** and **zz_x_side** in the **Surfaces** selection list.
- e. Disable the Auto Range and Draw Mesh options.
- f. Enter 413 K for **Min** and 473.15 K for **Max**.
- g. Click **Display**, and rotate the view as shown in Figure 7.6: Contours of Wall Temperature (p. 328).

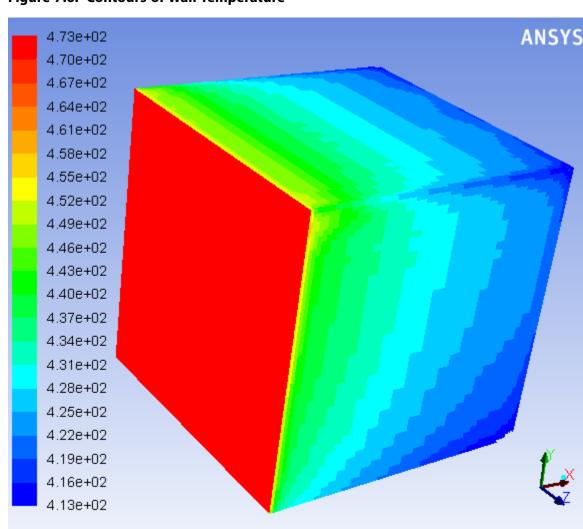


Figure 7.6: Contours of Wall Temperature

Contours of Wall Temperature (k)

ANSYS Fluent (3d, pbns, lam)

4. Display contours of radiation heat flux.

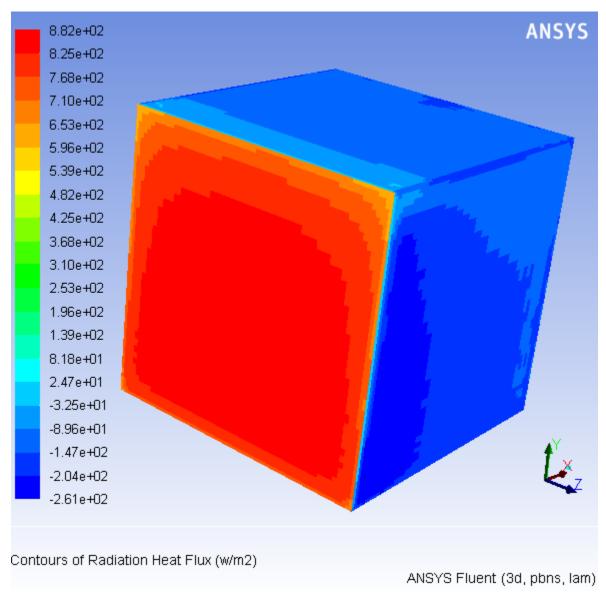
Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

Contours	
Options	Contours of
Filled	Wall Fluxes
Global Range	Radiation Heat Flux 🔻
Auto Range	Min Max
Clip to Range	0
Draw Mesh	Surfaces 🗵 🗏 🗏
	w-low-x
Levels Setup	w-low-y w-low-z
20 1	zz_center_z
	zz_x_side 👻
Surface Name Pattern	New Surface
Match	Surface Types 🔋 🗏 🖃
	axis
	dip-surf exhaust-fan
	fan -
	· · · · · · · · · · · · · · · · · · ·
Display	Compute Close Help

- a. Ensure that the **Filled** option is enabled in the **Options** group box.
- b. Select Wall Fluxes... and Radiation Heat Flux from the Contours of drop-down list.
- c. Make sure that all surfaces except **default-interior** and **zz_x_side** are selected in the **Surfaces** selection list.
- d. Click Display.
- e. Close the **Contours** dialog box.

Figure 7.7: Contours of Radiation Heat Flux (p. 330) shows the radiating wall (w-low-x) with positive heat flux and all other walls with negative heat flux.





5. Display vectors of velocity magnitude.

Graphics and Animations → \blacksquare Vectors → Set Up...

<i>.</i>			
Vectors			×
Options	Vectors of		
🔽 Global Range	Velocity		•
V Auto Range	Color by		
Clip to Range	Velocity		
Draw Mesh	Velocity Magnitude		•
Style	Min	Max	
arrow	0	0	
Scale Skip	Surfaces		
7 0 🛋	w-high-z		*
Vector Options	w-low-x		
	w-low-y w-low-z		
Custom Vectors	zz_center_z		=
	zz_x_side		-
Surface Name Pattern			
Match	New Surface 🔻		
	Surface Types		
	axis		
	clip-surf		
	exhaust-fan		
	fan -		*
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. 332).
- g. Close the **Vectors** dialog box.

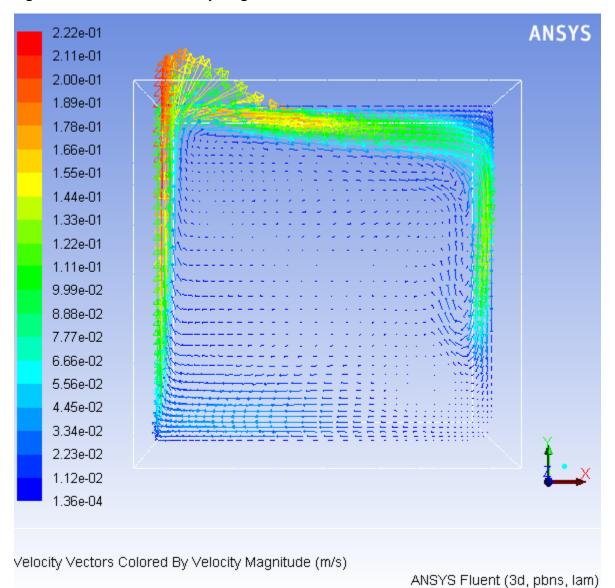


Figure 7.8: Vectors of Velocity Magnitude

6. Compute view factors and radiation emitted from the front wall (w-high-x) to all other walls.
 In the main menu, select Report → S2S Information...

S2S Information		X
Report Options View Factors Incident Radiation Boundary Types Report Options Exhaust-fan Inlet-vent Intake-fan Mass-flow-inlet Outflow INTER	From E =	To W-high-x W-high-y W-high-z W-low-x W-low-y W-low-y W-low-z
Compute Write	. Close	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.

7. Compute the total heat transfer rate.

```
 \mathbf{O} \mathsf{Reports} \to \mathbf{Fluxes} \to \mathsf{Set} \mathsf{Up...}
```

Flux Reports	×
 Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types B = 	Boundaries : Results default-interior w-high-x w-high-y w-low-x w-low-y w-low-z Net Results (w) -12.60360431671143 -12.64836502075195 -12.82371520996094 63.04367065429688 -12.16956233978272 -12.82330417633057 Net Results (w) -0.02488041
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.

The energy imbalance is approximately 0.04%.

8. Compute the total heat transfer rate for **w-low-x**.

 $\clubsuit Reports \rightarrow \blacksquare Fluxes \rightarrow Set Up...$

E Flux Reports		—
Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate	Boundaries default-interior w-high-x w-high-y w-high-z	Results 63.04367065429688
Boundary Types	w-low-x w-low-y w-low-z	
Boundary Name Pattern Match Save Output Parameter		
Compute Write	Close He	elp

- 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**.

The net heat load is approximately 63 W.

- 9. Compute the radiation heat transfer rate.
 - $\mathbf{O} \mathsf{Reports} \to \mathbf{Fluxes} \to \mathsf{Set} \mathsf{Up...}$

Flux Reports		—
 Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types	Boundaries default-interior w-high-x w-high-y w-high-z w-low-x w-low-y w-low-z	Results -10.81357002258301 -7.014934539794922 -10.53392601013184 51.42041778564453 -12.53998470306397 -10.53179740905762 Image: Note Results (w) -0.0137949
Compute Write	Close H	elp

- 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.

The heat imbalance is approximately -0.014 W.

10. Compute the radiation heat transfer rate for **w-low-x**.

 $\clubsuit Reports \rightarrow \blacksquare Fluxes \rightarrow Set Up...$

E Flux Reports		
Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate	Boundaries 🖹 🔳 🚍 default-interior w-high-x w-high-y w-high-z	Results
Boundary Types	w-low-x w-low-y w-low-z	
Match Save Output Parameter		
Compute Write	Close He	elp

- 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.

- 11. Display the temperature profile for the side wall.
 - $\textcircled{Plots} \rightarrow \overleftarrow{\blacksquare} XY Plot \rightarrow Set Up...$

Solution XY Plot		
Options Image: Option of Values Image: Option of Values </td <td>Plot Direction X 1 Y 0 Z 0 Load File Free Data</td> <td>Y Axis Function Temperature Wall Temperature X Axis Function Direction Vector Surfaces W-high-y W-high-z W-high-z W-how-x W-low-z zz_center_z zz_x_side New Surface New Surface</td>	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Temperature Wall Temperature X Axis Function Direction Vector Surfaces W-high-y W-high-z W-high-z W-how-x W-low-z zz_center_z zz_x_side New Surface New Surface
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. 339)).
- 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.

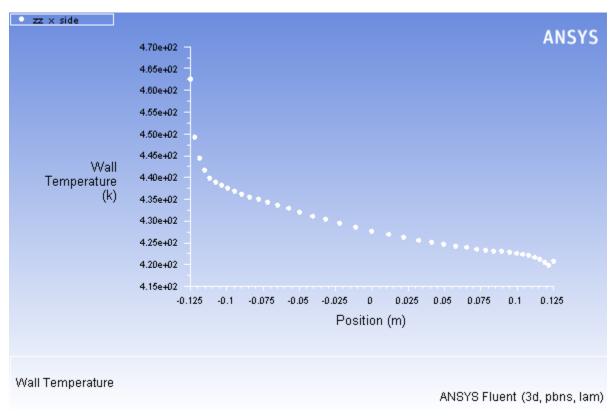


Figure 7.9: Temperature Profile Along the Outer Surface of the Box

12. Save the case and data files (rad_b_1.cas.gz and rad_b_1.dat.gz).

File \rightarrow Write \rightarrow Case & Data...

7.4.9. Comparing the Contour Plots after Varying Radiating Surfaces

1. Increase the number of faces per cluster to 10.

- 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. In the Solution Initialization task page, click Initialize.

Solution Initialization

3. Start the calculation by requesting 300 iterations.

CRun Calculation

The solution will converge in approximately 280 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 11.a 11.g of Postprocessing (p. 324), 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, in the manner described in step 3 of Postprocessing (p. 324).

• Graphics and Animations $\rightarrow \equiv$ Contours \rightarrow Set Up...

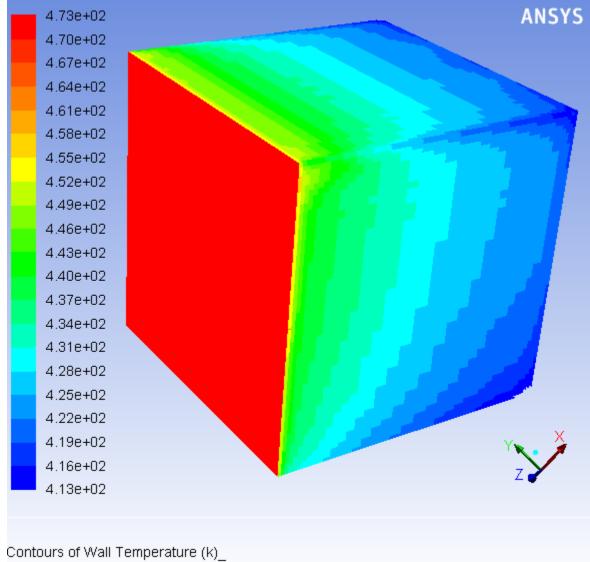


Figure 7.10: Contours of Wall Temperature: 1 Face per Surface Cluster

ANSYS Fluent (3d, pbns, lam)

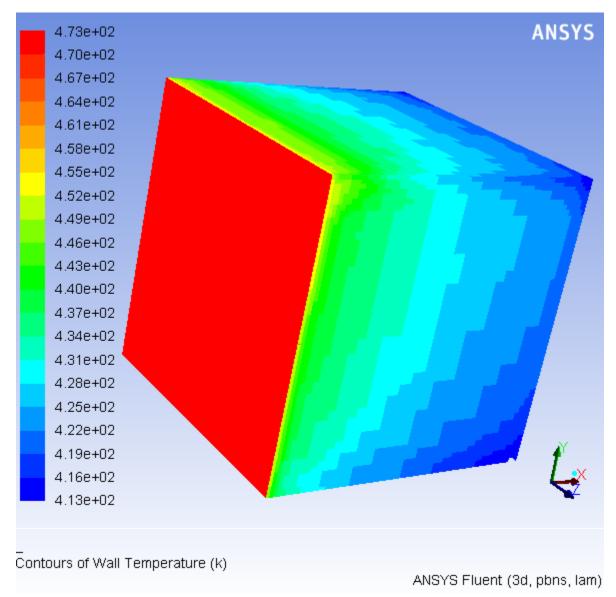


Figure 7.11: Contours of Wall Temperature: 10 Faces per Surface Cluster

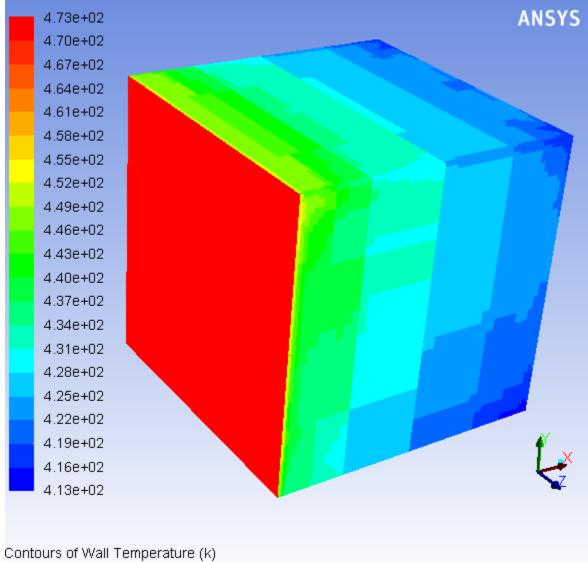


Figure 7.12: Contours of Wall Temperature: 100 Faces per Surface Cluster

ANSYS Fluent (3d, pbns, lam)

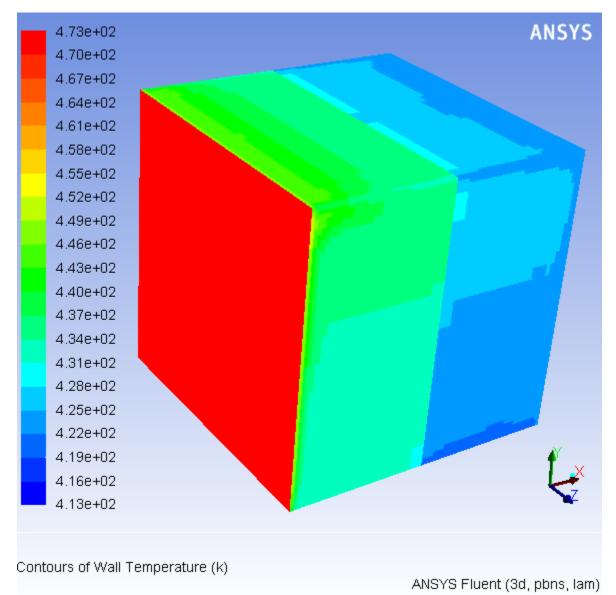


Figure 7.13: Contours of Wall Temperature: 400 Faces per Surface Cluster

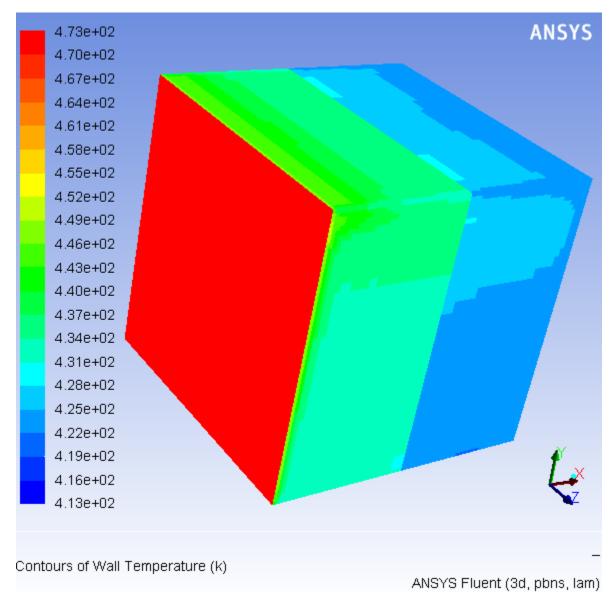


Figure 7.14: Contours of Wall Temperature: 800 Faces per Surface Cluster

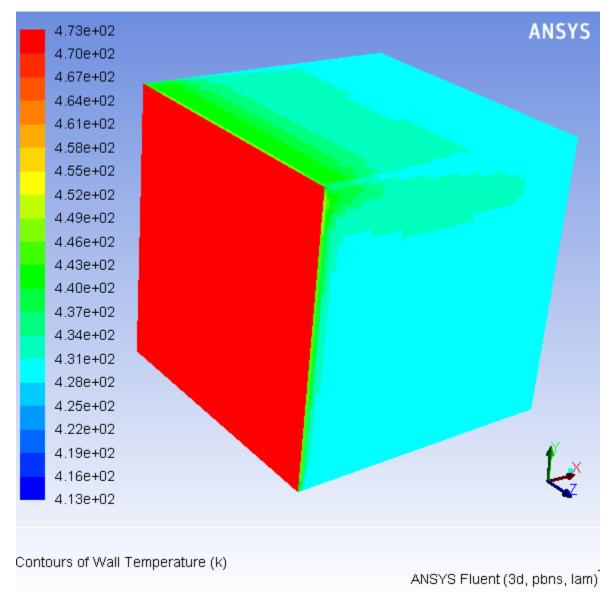


Figure 7.15: Contours of Wall Temperature: 1600 Faces per Surface Cluster

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. 348)).

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

Contours	×	
Options	Contours of	
V Filled	Radiation 👻	
Node Values Global Range	Surface Cluster ID 🗸	
Auto Range	Min Max	
Clip to Range	0 5	
Draw Profiles	Surfaces	
E Draw Mean	w-low-x	
Levels Setup	w-low-x w-low-y	
	w-low-z	
20 1		
	zz_x_side 👻	
Surface Name Pattern	New Surface 🔻	
Match	Surface Types	
	axis	
	clip-surf	
	exhaust-fan	
Display Compute Close Help		

- a. Ensure that the **Filled** option is enabled in the **Options** group box.
- b. Ensure that the **Node Values** option is disabled.
- c. Select Radiation... and Surface Cluster ID from the Contours of drop-down lists.
- d. Ensure that all surfaces except **default-interior** and **zz_x_side** are selected in the **Surfaces** selection list.
- e. Click **Display** and rotate the view as shown in Figure 7.16: Contours of **Surface Cluster ID**—1600 Faces per Surface Cluster (FPSC) (p. 348).
- f. Close the **Contours** dialog box.

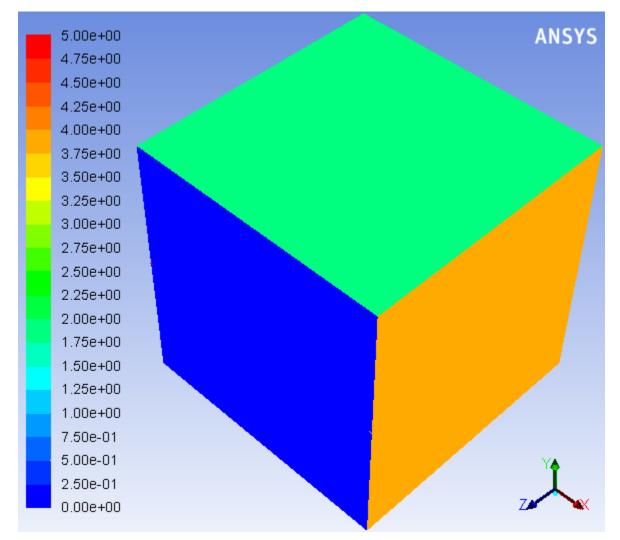


Figure 7.16: Contours of Surface Cluster ID—1600 Faces per Surface Cluster (FPSC)

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. 349)).

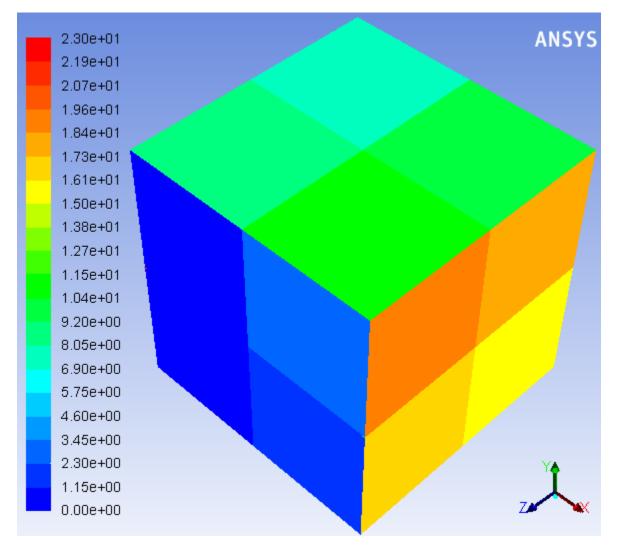




Figure 7.17: Contours of **Surface Cluster ID**—400 FPSC (p. 349) shows contours of **Surface Cluster ID** for 400 FPSC. This case shows better clustering compared to all of the other cases.

10. Create a plot that compares the temperature profile plots for 1, 10, 100, 400, 800, and 1600 FPSC.

$$\mathbf{O} \mathsf{Plots} \to \mathbf{File} \to \mathsf{Set} \mathsf{Up...}$$

- 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. 324).
 - ii. Click **OK** to close the **Select File** dialog box.
- b. Change the legend entry for the data series.

E File XY Plot	×
Plot Title	Legend Title 1
	Faces/Cluster
Files	Legend Entries
F:\H1_tutorial_re-runs\Tutorial 7\radiation_natural_c	Wall Temperature
	Add
	11 Delete
F:\H1_tutorial_re-runs\Tutorial 7\radiation_natural_d	1 Change Legend Entry
Plot Axes	Curves Close Help

- 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 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 Weight 1
	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. 351)) and close the **File XY Plot** dialog box.

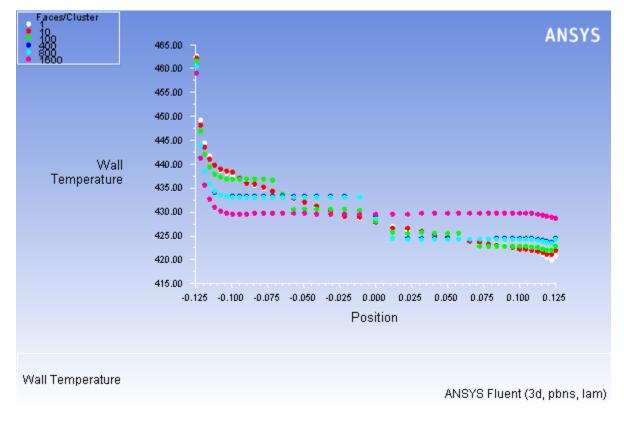


Figure 7.18: A Comparison of Temperature Profiles along the Outer Surface of the Box

7.4.10. 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 $({\tt rad_b_l.cas.gz}).$

```
\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Case...}
```

2. Set the partial enclosure parameters for the S2S model.

```
\textcircled{}{}^{\bullet} Boundary \ Conditions \rightarrow \fbox{}^{\bullet} w-low-x \rightarrow Edit...
```

S Wall	
Zone Name	
w-low-x	
Adjacent Cell Zone	
fluid	
Momentum Thermal Radiation Species DPM Multiphase	UDS Wall Film
S2S Parameters	
Faces Per Surface Cluster 1	
Participates in View Factor Calculation	
OK	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.

$\clubsuit Models \rightarrow \blacksquare Radiation \rightarrow Edit...$

- 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.

Participating Boundary Zones
Maximum Distance (m) from Critical Zone
To All Other Zones 0 Compute
To Participating Zones 0 Apply
Participating Boundary Zones 🖹 🗐 Non-Participating Boundary Zones 🗎 🗐
w-high-z w-low-x
w-low-z w-high-y
w-low-y
w-high-x
-> <-
Display Zones
Non-Participating Boundary Zones Temperature (k) 473.15
OK Cancel Help

- 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.

Note

You should compute the view factors only after you have specified the boundaries that will participate in the radiation model using the **Boundary Conditions** dialog box. If you first compute the view factors and then make a change to the boundary conditions, ANSYS Fluent will use the view factor file stored previously for calculating a solution, in which case, the changes that you made to the model will not be used for the calculation. Therefore, you should recompute the view factors and save the case file whenever you modify the number of objects that will participate in radiation.

- 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. In the Solution Initialization task page, click Initialize.

Solution Initialization

5. Start the calculation by requesting 300 iterations.

Run Calculation

The solution will converge in approximately 275 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.

$\clubsuit Reports \rightarrow \blacksquare Fluxes \rightarrow Set Up...$

Flux Reports		×
 Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types	Boundaries default-interior w-high-x w-high-y w-high-z w-low-x w-low-y w-low-z	Results -11.02793598175049 -7.330778121948242 -10.75697708129883 0 -12.6909990310669 -10.75475025177002 ▲ Ш ▶ Net Results (w) -52.56144
Compute Write	Close H	elp

- 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. 324), 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 $tp_1.xy$.

- 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 11 of Postprocessing (p. 324).
- 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.



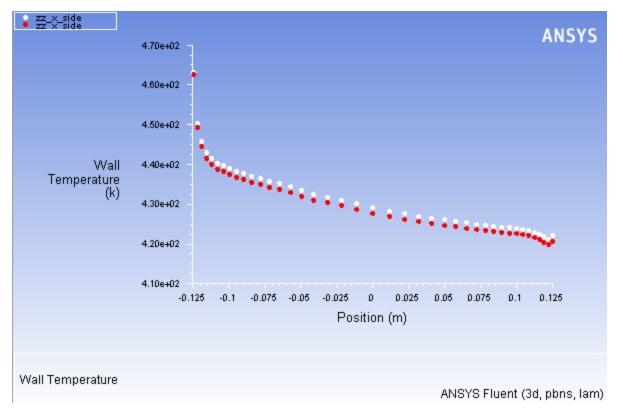


Figure 7.19: Wall Temperature Profile Comparison (p. 355) further confirms that the use of a partial enclosure did not significantly affect the results.

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 Modeling Radiation in the 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. 123).

Chapter 8: Using the Discrete Ordinates Radiation Model

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

This tutorial illustrates the set up and solution of flow and thermal modelling of a headlamp. The discrete ordinates (DO) radiation model will be used to model the radiation.

This tutorial demonstrates how to do the following:

- Read an existing mesh file into ANSYS Fluent.
- Set up the DO radiation model.
- · Set up material properties and boundary conditions.
- Solve for the energy and flow equations.
- Initialize and obtain a solution.
- Postprocess the resulting data.
- Understand the effect of pixels and divisions on temperature predictions and solver speed.

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. 123)

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

8.3. Problem Description

The problem to be considered is illustrated in Figure 8.1: Schematic of the Problem (p. 358), showing a simple two-dimensional section of a headlamp construction. The key components to be included are the bulb, reflector, baffle, lens, and housing. For simplicity, the heat output will only be considered from the bulb surface rather than the filament of the bulb. The radiant load from the bulb will cover all thermal radiation—this includes visible (light) as well as infrared radiation.

The ambient conditions to be considered are quiescent air at 20°C. Heat exchange between the lamp and the surroundings will occur by conduction, convection and radiation. The rear reflector is assumed to be well insulated and heat losses will be ignored. The purpose of the baffle is to shield the lens from direct radiation. Both the reflector and baffle are made from polished metal having a low emissivity and mirror-like finish; their combined effect should distribute the light and heat from the bulb across the lens. The lens is made from glass and has a refractive index of 1.5.

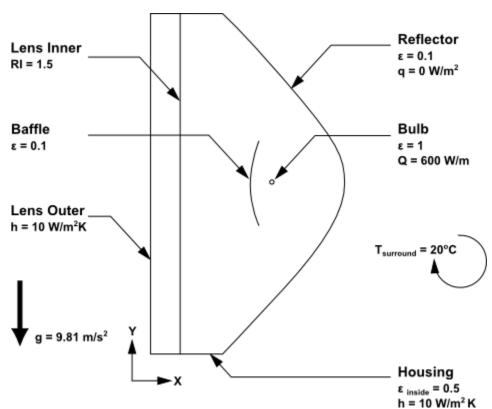


Figure 8.1: Schematic of the Problem

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

- 8.4.10. Iterate for Higher Pixels
- 8.4.11. Iterate for Higher Divisions
- 8.4.12. Make the Reflector Completely Diffuse
- 8.4.13. Change the Boundary Type of Baffle

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 15.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip do_rad_R150.zip to your working folder.

The mesh file do.msh.gz can be found in the do_rad 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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Run in single precision (disable Double Precision).
- 11. Ensure that Serial is selected under Processing Options.

8.4.2. Mesh

1. Read the mesh file do.msh.gz.

File \rightarrow Read \rightarrow Mesh...

As the mesh file is read, ANSYS Fluent will report the progress in the console.

8.4.3. General Settings

1. Check the mesh.

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.

2. Scale the mesh.

$\clubsuit General \rightarrow Scale...$

Scale Mesh	—
Domain Extents	Scaling
Xmin (mm) -37.5 Xmax (mm) 49.99738	 Convert Units Specify Scaling Factors
Ymin (mm) -87.30001 Ymax (mm) 87.96	Mesh Was Created In
View Length Unit In mm	Scaling Factors X 0.001 Y 0.001 Scale Unscale
Close Help	

a. Select **mm** from the **View Length Unit In** drop-down list.

The **Domain Extents** will be reported in **mm**.

- b. Select mm from the Mesh Was Created In drop-down list.
- c. Click Scale and close the Scale Mesh dialog box.
- 3. Check the mesh.



Note

It is good practice to check the mesh after manipulating it (scale, convert to polyhedra, merge, separate, fuse, add zones, or smooth and swap).

4. Examine the mesh.

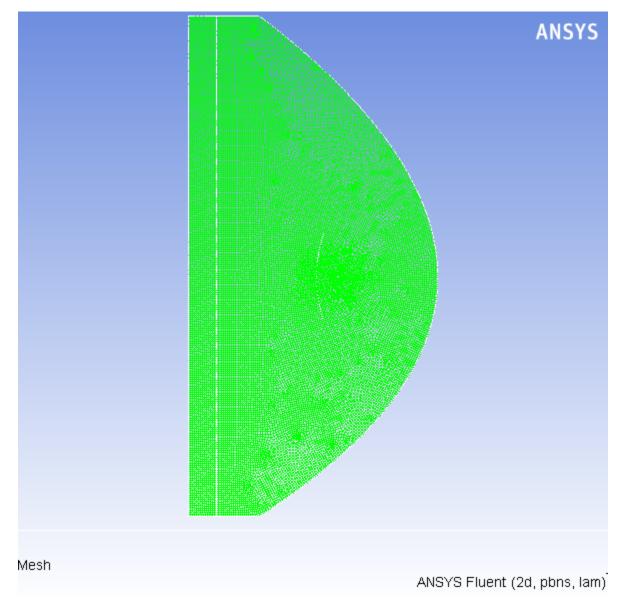


Figure 8.2: Graphics Display of Mesh

5. Change the unit of temperature to centigrade.

 $\textcircled{} General \rightarrow Units...$

- a. Select temperature from the Quantities selection list.
- b. Select **c** from the **Units** selection list.
- c. Close the Set Units dialog box.
- 6. Confirm the solver settings and enable gravity.

⇔General

General		
Mesh		
Scale Display	Check	eport Quality
Solver		
Type Pressure-Based Density-Based	Velocity Formu Absolute Relative	lation
Time ◉ Steady ◯ Transient	2D Space Planar Axisymmetric Axisymmetric Swirl	
Gravity		Units
Gravitational Acceleratio	on	
X (m/s2) 0	(Đ
Y (m/s2) -9.81		P
Z (m/s2)		P
-		
Help		

- a. Retain the default selections in the **Solver** group box.
- b. Enable **Gravity**.
- c. Enter -9.81 m/s^2 for **Gravitational Acceleration** in the **Y** direction.

8.4.4. Models

1. Enable the energy equation.

�Models →		Energy →	Edit
-----------	--	----------	------

Energy	×
Energy	
OK Cancel	Help

2. Enable the **DO** radiation model.

Radiation Model				×
Model Off Rosseland P1 Discrete Transfer (DTRM) Surface to Surface (S2S) Discrete Ordinates (DO) DO/Energy Coupling	Iteration Parameters Energy I Angular Discretization Theta Divisions 2 Phi Divisions 2 Theta Pixels 1 Phi Pixels 1		ions per Radiation Iteration 1 Non-Gray Model Number of Bands 0	
	OK Cancel	He	lp	

a. Select Discrete Ordinates (DO) in the Model list.

The **Radiation Model** dialog box expands to show the related inputs.

b. Set the Energy Iterations per Radiation Iteration to 1.

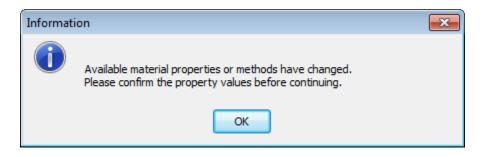
As radiation will be the dominant mode of heat transfer, it is beneficial to reduce the interval between calculations. For this small 2D case we will reduce it to 1.

c. Retain the default settings for **Angular Discretization**.

d. Click **OK** to close the **Radiation Model** dialog box.

An Information dialog box will appear, indicating that material properties have changed.

e. Click **OK** in the **Information** dialog box.



8.4.5. Materials

1. Set the properties for **air**.

Materials →	air → Create/Edit
-------------	-------------------

Create/Edit Materials		—
Name air Chemical Formula	Material Type fluid	Order Materials by Name Chemical Formula
	FLUENT Fluid Materials	FLUENT Database User-Defined Database
Properties		
Density (kg/m3)	incompressible-ideal-gas	
Cp (Specific Heat) (j/kg-k)	constant Edit Edit	
Thermal Conductivity (w/m-k)	constant Edit Edit	
Viscosity (kg/m-s)	constant Edit 1.789%e-05	
	Change/Create Delete Close Help	

a. Select incompressible-ideal-gas from the Density drop-down list.

Since pressure variations are insignificant compared to temperature variation, we choose incompressible-ideal-gas law for density.

- b. Retain the default settings for all other parameters.
- c. Click Change/Create and close the Create/Edit Materials dialog box.

2. Create a new material, lens.

Create/Edit Materials			N		
ame lens		Material Type solid	ß		Order Materials by
Chemical Formula		Fluent Solid Mate	rials		Chemical Formula
		Mixture			User-Defined Database
roperties					
Absorption Coefficient (1/m)	constant		▼ Edit	ŕ	
	200				
Scattering Coefficient (1/m)	constant		• Edit		
	0				
Scattering Phase Function	isotropic		• Edit		
				Е	
Refractive Index constant			▼ Edit		
	1.5				

- a. Enter lens for Name and delete the entry in the Chemical Formula field.
- b. Enter 2200 Kg/m^3 for **Density**.
- c. Enter 830 J/Kg-K for Cp (Specific Heat).
- d. Enter 1.5 W/m-K for **Thermal Conductivity**.
- e. Enter 200 1/m for Absorption Coefficient.
- f. Enter 1.5 for **Refractive Index**.
- g. Click Change/Create.

A **Question** dialog box will open, asking if you want to overwrite **aluminum**.

Question	
?	Change/Create mixture and Overwrite aluminum?
	Yes No

h. Click **No** in the **Question** dialog box to retain **aluminum** and add the new material (**lens**) to the materials list.

The **Create/Edit Materials** dialog box will be updated to show the new material, **lens**, in the **ANSYS Fluent Solid Materials** drop-down list.

i. Close the Create/Edit Materials dialog box.

8.4.6. Cell Zone Conditions

Cell Zone Conditions

Cell Zone Con	ditions	
Zone		
fluid		
lens		
Phase mixture	Type fluid •	ID 17
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh		
Porous Formulation Operation Superficial Velo		
 Physical Velocit 		
Help		

1. Ensure that **air** is selected for fluid.



🖬 Fluid 🗾				
Zone Name				
fluid				
Material Name air				
Frame Motion Source Terms				
Mesh Motion Fixed Values Participates In Radiation				
Porous Zone				
Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Multiphase				
Rotation-Axis Origin				
X (mm) 0 constant				
Y (mm) 0 constant				
Ψ				
OK Cancel Help				

- a. Retain the default selection of **air** from the **Material Name** drop-down list.
- b. Click **OK** to close the **Fluid** dialog box.
- 2. Set the cell zone conditions for the **lens**.

 $\textcircled{Cell Zone Conditions} \rightarrow \overleftarrow{\sqsubseteq} \text{ lens} \rightarrow \text{ Edit...}$

Solid 💌
Zone Name
lens
Material Name lens
Frame Motion Source Terms Participates In Radiation
Mesh Motion Fixed Values
Reference Frame Mesh Motion Source Terms Fixed Values
Rotation-Axis Origin
X (mm) 0 constant
Y (mm) 0 constant
-
OK Cancel Help

- a. Select lens from the Material Name drop-down list.
- b. Enable Participates In Radiation.
- c. Click **OK** to close the **Solid** dialog box.

8.4.7. Boundary Conditions

Boundary Conditions

Boundary Conditions

Zone		
air-interior		
baffle		
baffle-shadow		
bulb-outer		
housing		
lens-inner		
lens-inner-shadow	v	
lens-interior		
lens-outer		
lens-sides		
reflector		
<u> </u>		
Phase	Туре	ID
mixture	→ wall →	24
Edit	Copy Profiles	
	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh	Periodic Conditions	
Help		

1. Set the boundary conditions for the **baffle**.



💶 Wall			
Zone Name			
baffle			
Adjacent Cell Zone			
fluid			
Shadow Face Zone			
baffle-shadow			
Momentum Thermal	Radiation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
Heat Flux	Internal Emissivity	0.1	constant 👻
 Temperature Coupled 		Wall Thickness (mr	m) 0 (P
Material Name	Heat Generation Rate (w/m3	0	constant 👻
aluminum	▼ Edit	. [constant v
	OK	el Help	

- a. Click the **Thermal** tab and enter 0.1 for **Internal Emissivity**.
- b. Click the Radiation tab and enter 0 for Diffuse Fraction.
- c. Click **OK** to close the **Wall** dialog box.
- 2. Set the boundary conditions for the **baffle-shadow**.

 $\textcircled{Boundary Conditions} \rightarrow \fbox{baffle-shadow} \rightarrow \texttt{Edit...}$

🛃 Wall	×
Zone Name	
baffle-shadow	
Adjacent Cell Zone	
fluid	
Shadow Face Zone	_
baffle	
Momentum Thermal Radiation Species DPM Multiphase	UDS Wall Film
Thermal Conditions	
Heat Flux Internal Emission	vity 0.1 constant
Temperature Coupled	Wall Thickness (mm)
Material Name Heat Generation Rate (w/r	
aluminum V Edit	m3) 0 constant •
OK Ca	incel Help

- a. Click the **Thermal** tab and enter 0.1 for **Internal Emissivity**.
- b. Click the Radiation tab and enter 0 for Diffuse Fraction.
- c. Click **OK** to close the **Wall** dialog box.
- 3. Set the boundary conditions for the **bulb-outer**.

 $\textcircled{P} Boundary Conditions \rightarrow \fbox{bulb-outer} \rightarrow \texttt{Edit...}$

💶 Wall			—
Zone Name			
bulb-outer			
Adjacent Cell Zone			
fluid			
Momentum Thermal Ra	diation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
Heat Flux	Heat Flux (w/m2	150000	constant 👻
 Temperature Convection 	Internal Emissivit	y 1	constant 👻
Radiation		Wall Thickness (n	
 Mixed via System Coupling 		waii micchess (ii	nm) 0 E
Material Name	Heat Generation Rate (w/m3	0	constant 🔹
aluminum	▼ Edit		
	OK	el Help	

a. Click the **Thermal** tab and enter 150000 W/m^2 for **Heat Flux**.

The circumference of **bulb-outer** is approximately 0.004 m. Therefore the 600 W/m lineal heat flux specified in the problem description corresponds to 150000 W/m^2 .

- b. Retain the value of 1 for Internal Emissivity.
- c. Click **OK** to close the **Wall** dialog box.
- 4. Set the boundary conditions for the **housing**.

 $\clubsuit Boundary \ Conditions \rightarrow \fbox housing \rightarrow Edit...$

💶 Wall			—
Zone Name			
housing			
Adjacent Cell Zone			
fluid			
Momentum Thermal Rad	diation Species DPM Multiphase UDS	Wall Film	
Thermal Conditions			
Heat Flux	Heat Transfer Coefficient (w/m2-k)	10	constant 👻
Convection	Free Stream Temperature (c)	20	constant 👻
 Radiation Mixed via System Coupling 	External Emissivity	1	constant 💌
Material Name	External Radiation Temperature (c)	20	constant 💌
aluminum	Edit Internal Emissivity	0.5	constant 👻
		Wall Thickness (m	nm) 0 P
	Heat Generation Rate (w/m3)	0	constant 💌
L			
	OK Cancel	Help	

- a. Click the Thermal tab and select Mixed in the Thermal Conditions group box.
- b. Enter 10 W/m^2 -K for Heat Transfer Coefficient.
- c. Enter 20 C for Free Stream Temperature.
- d. Retain the value of 1 for **External Emissivity**.
- e. Enter 20 C for External Radiation Temperature.
- f. Enter 0.5 for Internal Emissivity.
- g. Click **OK** to close the **Wall** dialog box.
- 5. Set the boundary conditions for the **lens-inner**.

$\textcircled{}Boundary \text{ Conditions} \rightarrow \fbox{} lens-inner \rightarrow \texttt{Edit...}$

The inner and outer surface of the lens will be set to semi-transparent conditions. This allows radiation to be transmitted through the wall between the two adjacent participating cell zones. It also calculates the effects of reflection and refraction at the interface. These effects occur because of the change in refractive index (set through the material properties) and are a function of the incident angle of the radiation and the surface finish. In this case, the lens is assumed to have a very smooth surface so the diffuse fraction will be set to 0.

On the internal walls (wall/wall-shadows) it is important to note the adjacent cell zone: this is the zone the surface points into and may influence the settings on the diffuse fraction (these can be different on both sides of the wall).

💽 Wall	
Zone Name	
lens-inner	
Adjacent Cell Zone	
fluid	
Shadow Face Zone	
lens-inner-shadow	
Momentum Thermal Radiation Species DPM Multiphase U	DS Wall Film
BC Type	
semi-transparent	
Diffuse Fraction	
OK Cancel	Help

- a. Click the **Radiation** tab.
- b. Select semi-transparent from the BC Type drop-down list.
- c. Enter 0 for Diffuse Fraction.
- d. Click **OK** to close the **Wall** dialog box.
- 6. Set the boundary conditions for the **lens-inner-shadow**.

• Boundary Conditions $\rightarrow \equiv$ lens-inner-shadow \rightarrow Edit...

- a. Click the Radiation tab.
- b. Retain the selection of **semi-transparent** from the **BC Type** drop-down list.
- c. Enter 0 for Diffuse Fraction.
- d. Click **OK** to close the **Wall** dialog box.
- 7. Set the boundary conditions for the **lens-outer**.

$\clubsuit Boundary Conditions \rightarrow \blacksquare Iens-outer \rightarrow Edit...$

The surface of the lamp cools mainly by natural convection to the surroundings. As the outer lens is transparent it must also lose radiation to the surroundings, while the surroundings will supply a small source of background radiation associated with the temperature. For the lens, a semi-transparent condition

is used on the outside wall. A mixed thermal condition provides the source of background radiation as well as calculating the convective cooling on the outer lens wall. For a semi-transparent wall, the source of background radiation is added directly to the DO radiation rather than to the energy equation; an external emissivity of 1 is used, in keeping with the assumption of a small object in a large enclosure. As the background radiation is supplied from the thermal conditions, there is no need to supply this as a source of irradiation under the **Radiation** tab for the wall boundary condition. The only other setting required here is the surface finish of the outer surface of the lens; the diffuse fraction should be set to 0 as the lens is assumed to be smooth.

💶 Wall			
Zone Name			
lens-outer			
Adjacent Cell Zone			
lens			
Momentum Thermal Rad	diation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
🔘 Heat Flux	Heat Transfer Coefficient (w/r	m2-k) 10	constant 👻
 Temperature Convection 	Free Stream Temperatur	re (c) 20	constant 👻
 Radiation Mixed 	External Emis	sivity 1	constant 🗸
via System Coupling			constant 🔻
Material Name	External Radiation Temperatur	re (c) 20	constant 👻
aluminum	▼ Edit Internal Emis	sivity 1	constant 👻
		Wall Thickness (m	m) 0
	Heat Generation Rate (vi	v/m3) 0	constant 👻
	OK Can	cel Help	

- a. Click the **Thermal** tab and select **Mixed** in the **Thermal Conditions** group box.
- b. Enter 10 W/m^2 -K for Heat Transfer Coefficient.
- c. Enter 20 C for Free Stream Temperature.
- d. Retain the value of 1 for External Emissivity.

For a semi-transparent wall the internal emissivity has no effect as there is no absorption or emission on the surface. So the set value is irrelevant.

- e. Enter 20 C for External Radiation Temperature.
- f. Ensure that aluminum is selected from the Material Name drop-down list.

Because lens-outer is modeled as a zero-thickness wall, the choice of material is unimportant.

g. Click the **Radiation** tab.

💶 Wall	
Zone Name	
lens-outer	
Adjacent Cell Zone	
lens	
Momentum Thermal Radiation Sp	ecies DPM Multiphase UDS Wall Film
BC Type	Direct Irradiation
semi-transparent 💌	(w/m2) 0 constant
	Apply Direct Irradiation Parallel to the Beam
Beam Width	Diffuse Irradiation
Theta (deg) 1e-06	(w/m2) 0 constant
Phi (deg) 1e-06 P	-
Beam Direction	Diffuse Fraction
X 1 constant -	
Y 0 constant -	E
Constant V	
Z 0 constant -	
	OK Cancel Help

- h. Select semi-transparent from the BC Type drop-down list.
- i. Enter 0 for Diffuse Fraction.
- j. Click **OK** to close the **Wall** dialog box.
- 8. Set the boundary conditions for the **reflector**.

Like the baffles, the reflector is made of highly polished aluminum, giving it highly reflective surface property; about 90% of incident radiation reflects from this surface so only about 10% gets absorbed. Based on Kirchhoff's law, we can assume emissivity equals absorptivity. Therefore, we apply internal emissivity = 0.1. We also assume a clean reflector (diffuse fraction = 0).

- a. Click the **Thermal** tab and enter 0.1 for **Internal Emissivity**.
- b. Click the Radiation tab and enter 0 for Diffuse Fraction.
- c. Click OK to close the Wall dialog box.

8.4.8. Solution

1. Set the solution parameters.

CSolution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
SIMPLE	
Spatial Discretization	
Gradient	*
Least Squares Cell Based 🔹	
Pressure	
Body Force Weighted 🔹	
Momentum	
Second Order Upwind 🗸	
Energy	
Second Order Upwind 🗸	
Discrete Ordinates	
First Order Upwind 👻	-
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation Pseudo Transient	
Utab Ordan Tana Balawatian (
High Order Term Relaxation Options	
Default	
Help	

- a. Select **Body Force Weighted** from the **Pressure** drop-down list in the **Spatial Discretization** group box.
- 2. Set the under-relaxation factors.

♀Solution Controls

Inder-Relaxation Factors	_
Pressure	
0.7	
Density	
1	
Body Forces	Ε
1	
Momentum	
0.6	
Energy	
1	
	Ŧ
Default	
Equations Limits Advanced	
Help	

- a. Enter 0.7 for Pressure.
- b. Enter 0.6 for **Momentum**.
- 3. Reduce the convergence criteria.

Monitors →
 Eresiduals → Edit...

Residual Monitors					×
Options Print to Console Plot Window 1 Curves Axes	Equations Residual continuity x-velocity y-velocity	Monitor C	Check Convergence	Absolute Criteria 1e-4 0.001 0.001	
Iterations to Plot	energy Residual Values			1e-06 Convergence Cr	
Iterations to Store	Normalize		Iterations	absolute	•
	Scale	al Scale			
OK Plot Renormalize Cancel Help					

- a. Enter 1e-4 for Absolute Criteria for continuity.
- b. Ensure that **Print to Console** and **Plot** are enabled.
- c. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Initialize the solution.

Solution Initialization

Solution Initialization			
Initialization Methods			
 Hybrid Initialization Standard Initialization 			
More Settings Initialize			
Patch			
Reset DPM Sources Reset Statistics			
Help			

- a. Retain the selection of **Hybrid Initialization** from the **Initialization Methods** group box.
- b. Click Initialize.
- 5. Save the case file (do.cas.gz)

```
\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}
```

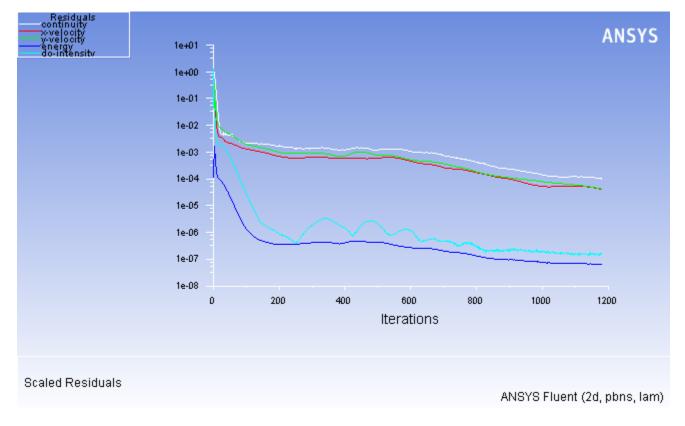
6. Start the calculation by requesting 1500 iterations.

CRun Calculation

Run Calculation	
Check Case Preview	Mesh Motion
Number of Iterations Reporting	Interval
Profile Update Interval	
Data File Quantities Acous	tic Signals
Calculate	
Help	

- a. Enter 1500 for Number of Iterations.
- b. Click Calculate.

Figure 8.3: Residuals



The solution will converge in approximately 1180 iterations.

7. Save the case and data files (do.cas.gz and do.dat.gz).

File \rightarrow Write \rightarrow Case & Data...

8.4.9. Postprocessing

1. Display velocity vectors.

Graphics and Animations → $\overline{\equiv}$ Vectors → Set Up...

_				
Vectors			×	
Options	Vectors of			
Global Range	Velocity		-	
V Auto Range	Color by			
✓ Clip to Range	Velocity 👻			
Auto Scale Draw Mesh				
Drawmesh	Velocity Magnitude		•	
Style	Min	Max		
arrow 🔻	0	0		
Scale Skip	Surfaces			
10 0	air-interior			
	baffle		<u>í</u>	
Vector Options	baffle-shadow		E	
Custom Vectors	bulb-outer			
Custom rectors	housing			
	lens-inner		-	
Surface Name Pattern	New Surface -			
Match	INEW SUITACE +			
	Surface Types			
	axis		*	
	clip-surf exhaust-fan			
	exnaust-ran fan			
			*	
Display	Compute Close	Help		

- a. Enter 10 for Scale.
- 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** (Figure 8.4: Vectors of Velocity Magnitude (p. 382)).
- e. Close the Vectors dialog box.

Tip

You may need to click the **Fit to Window** button to center the vector graphic in your graphics window.

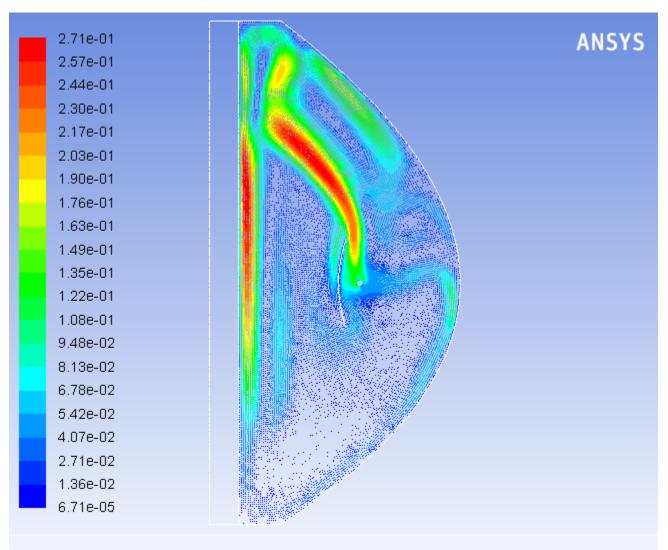


Figure 8.4: Vectors of Velocity Magnitude

Velocity Vectors Colored By Velocity Magnitude (m/s)

ANSYS Fluent (2d, pbns, lam)

2. Create the new surface, lens.

Surface \rightarrow Zone...

one	Surfaces
air-interior baffle baffle-shadow bulb-outer fluid housing	air-interior baffle baffle-shadow bulb-outer housing lens-inner
lens	lens-inner-shadow
lens-inner lens-inner-shadow lens-interior lens-outer lens-sides reflector	lens-interior New Surface Name lens

- a. Select lens from the Zone selection list.
- b. Click **Create** and close the **Zone Surface** dialog box.

\bigcirc Graphics and Animations $\rightarrow \stackrel{\frown}{\equiv}$ Contours \rightarrow Set Up...

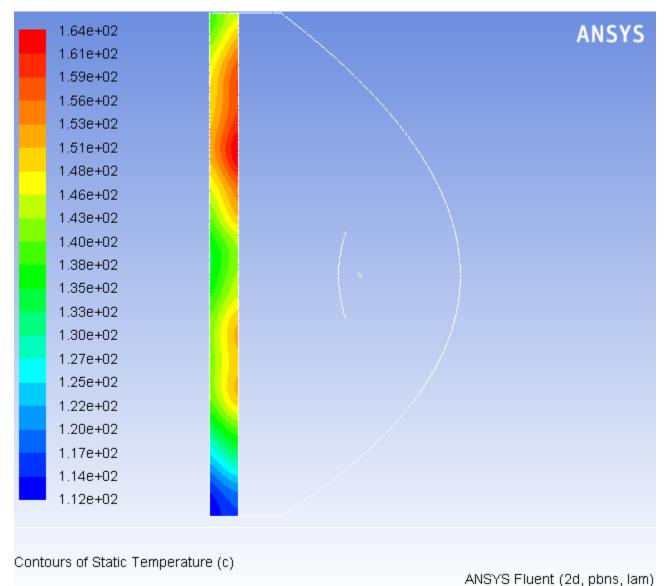
3. Display contours of static temperature.

Contours	×
Options	Contours of
V Filled	Temperature 👻
Vode Values	Static Temperature 🗸
V Auto Range	Min Max
Clip to Range	0 0
Draw Mesh	Surfaces
	baffle-shadow
Levels Setup	bulb-outer
	housing E
	lens-inner 🗸
Surface Name Pattern	New Surface 🕶
Match	Surface Types
	axis
	clip-surf
	exhaust-fan fan
Display	Compute Close Help

- a. Enable **Filled** in the **Options** group box.
- b. Disable **Global Range** in the **Options** group box.

- c. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- d. Select lens from the Surfaces selection list.
- e. Click **Display** (Figure 8.5: Contours of Static Temperature (p. 384)).

Figure 8.5: Contours of Static Temperature



- f. Close the **Contours** dialog box.
- 4. Display temperature profile for the **lens-inner**.

 $\textcircled{Plots} \rightarrow \fbox{XY Plot} \rightarrow \texttt{Set Up...}$

Solution XY Plot			- ×
Options Node Values Position on X Axis Position on Y Axis Write to File Order Points	Plot Direction X 0 Y 1 Z 0 Load File Free Data	Y Axis Function Direction Vector X Axis Function Temperature Wall Temperature Surfaces air-interior baffle baffle-shadow bulb-outer housing lens lens-inner New Surface ▼	
Plot	Axes	Curves Close Help	

- a. Disable both Node Values and Position on X Axis in the Options group box.
- b. Enable Position on Y Axis.
- c. Enter 0 and 1 for **X** and **Y**, respectively, in the **Plot Direction** group box.
- d. Retain the default selection of Direction Vector from the Y Axis Function drop-down list.
- e. Select Temperature... and Wall Temperature from the X Axis Function drop-down lists.
- f. Select lens-inner from the Surfaces selection list.
- g. Click the Axes... button to open the Axes Solution XY Plot dialog box.

Axes - Solution XY Plot		
Axis a X Y Label Temperature on Lens Inner	Number Format Type float Precision 0	Major Rules Color foreground Weight 1
Options Log Auto Range Major Rules Minor Rules	Range Minimum 0 Maximum 0	Minor Rules Color dark gray - Weight 1
	Apply Close Help	

- i. Ensure that **X** is selected in the **Axis** list.
- ii. Enter Temperature on Lens Inner for Label.
- iii. Select float from the Type drop-down list in the Number Format group box.
- iv. Set **Precision** to 0.
- v. Click Apply.

Axes - Solution XY Plot		— ×
Axis X Y Label Y Position on Lens Inner	Number Format Type float Precision 0	Major Rules Color foreground Weight 1
Options Log Auto Range Major Rules Minor Rules	Range Minimum 0 Maximum 0	Minor Rules Color dark gray Weight 1
	Apply Close Help	

vi. Select Y in the Axis list.

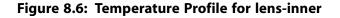
vii. Enter Y Position on Lens Inner for Label.

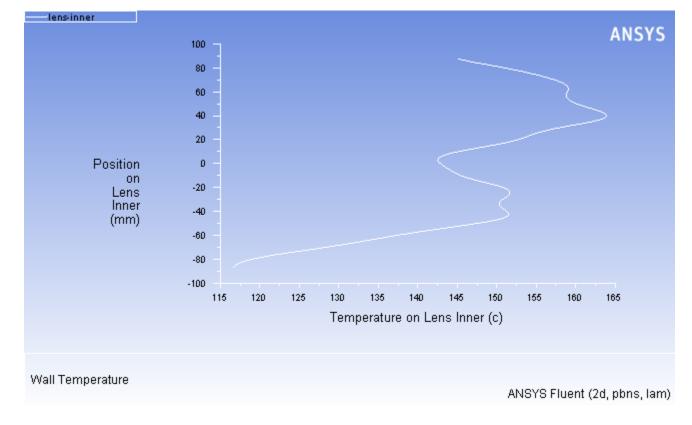
viii.Select float from the Type drop-down list in the Number Format group box.

- ix. Set Precision to 0.
- x. Click Apply and close the Axes Solution XY Plot dialog box.
- h. Click the Curves... button to open the Curves Solution XY Plot dialog box.

Curves -	Solution XY Plot	—	
Curve #	Line Style Pattern Color foreground Weight 1	Marker Style Symbol Color foreground Size 0.3	
Apply Close Help			

- i. Select the line pattern as shown in the Curves Solution XY Plot dialog box.
- ii. Select the symbol pattern as shown in the Curves Solution XY Plot dialog box.
- iii. Click Apply and close the Curves Solution XY Plot dialog box.
- i. Click Plot (Figure 8.6: Temperature Profile for lens-inner (p. 387)).





- j. Enable Write to File and click the Write... button to open the Select File dialog box.
 - i. Enter do_2x2_1x1.xy for XY File.
 - ii. Click **OK** to close the **Select File** dialog box.

k. Close the **Solution XY Plot** dialog box.

8.4.10. Iterate for Higher Pixels

1. Increase pixelation for accuracy.

$\textcircled{Models} \rightarrow \overleftarrow{E} Radiation \rightarrow Edit...$

For semi-transparent and reflective surfaces, increasing accuracy by increasing pixilation is more efficient than increasing theta and phi divisions.

Radiation Model		×
Model Off Rosseland P1 Discrete Transfer (DTRM) Surface to Surface (S2S) Discrete Ordinates (DO) DO/Energy Coupling	Iteration Parameters Flow 2 Angular Discretization Theta Divisions 2 Phi Divisions 2 Theta Pixels 2 Phi Pixels 2	W Iterations per Radiation Iteration 1
	OK Cancel	Help

- a. Set both Theta Pixels and Phi Pixels to 2.
- b. Click **OK** to close the **Radiation Model** dialog box.
- 2. Request 1500 more iterations.

Run Calculation

The solution will converge in approximately 500 additional iterations.

3. Save the case and data files (do_2x2_2x2_pix.cas.gz and do_2x2_2x2_pix.dat.gz).

File \rightarrow Write \rightarrow Case & Data...

4. Display temperature profile for the **lens-inner**.

- a. Disable Write to File.
- b. Retain the default settings and plot the temperature profile.
- c. Enable Write to File and click the Write... button to open the Select File dialog box.

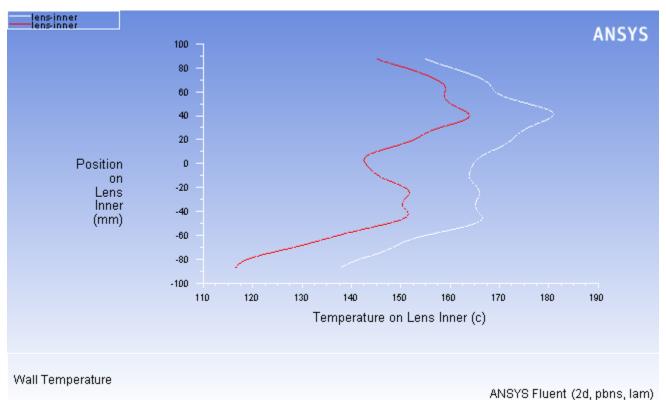
- i. Enter do_2x2_2x2_pix.xy for XY File.
- ii. Click OK to close the Select File dialog box.
- d. Click the Load File... button to open the Select File dialog box.
 - i. Select do_2x2_1x1.xy.
 - ii. Click **OK** to close the **Select File** dialog box.
- e. Click the Curves... button to open Curves Solution XY Plot dialog box.
 - i. Set Curve # to 1.
 - ii. Select the line pattern as shown in the **Curves Solution XY Plot** dialog box.
 - iii. Select the symbol pattern as shown in the Curves Solution XY Plot dialog box.

Curves -	Solution XY Plot	×
Curve #	Line Style Pattern Color red Weight 1	Marker Style Symbol Color red Size 0.3
	Apply Close	Help

iv. Click Apply and close the Curves - Solution XY Plot dialog box.

- f. Disable Write to File.
- g. Click Plot (Figure 8.7: Temperature Profile for lens-inner (p. 390)).





- h. Close the Solution XY Plot dialog box.
- 5. Increase both Theta Pixels and Phi Pixels to 3 and continue iterations.

$$\bigcirc$$
 Models $\rightarrow \blacksquare$ Radiation \rightarrow Edit...

6. Click the **Calculate** button.

CRun Calculation

The solution will converge in approximately 450 additional iterations.

7. Save the case and data files (do_2x2_3x3_pix.cas.gz and do_2x2_3x3_pix.dat.gz).

```
File \rightarrow Write \rightarrow Case & Data...
```

8. Display temperature profile for the **lens-inner**.

```
 \textcircled{Plots} \rightarrow \overleftarrow{\blacksquare} XY Plot \rightarrow Set Up...
```

- a. Ensure that Write to File is disabled.
- b. Ensure that all files are deselected from the File Data selection list.
- c. Ensure that lens-inner is selected from the Surfaces selection list.
- d. Click Plot.

- e. Enable Write to File and save the file as do_2x2_3x3_pix.xy.
- 9. Repeat the procedure for 10 Theta Pixels and Phi Pixels and save the case and data files (do_2x2_10x10_pix.cas.gz and do_2x2_10x10_pix.dat.gz).
 - a. Save the file as do_2x2_10x10_pix.xy.
- 10. Read in all the files and plot them.

```
 \textcircled{Plots} \rightarrow \overleftarrow{\sqsubseteq} XY Plot \rightarrow Set Up...
```

- a. Disable Write to File.
- b. Click the Load File... button to open the Select File dialog box.
 - i. Select all the xy files and close the **Select File** dialog box.

Note

Selected files will be listed in the XY File(s) selection list.

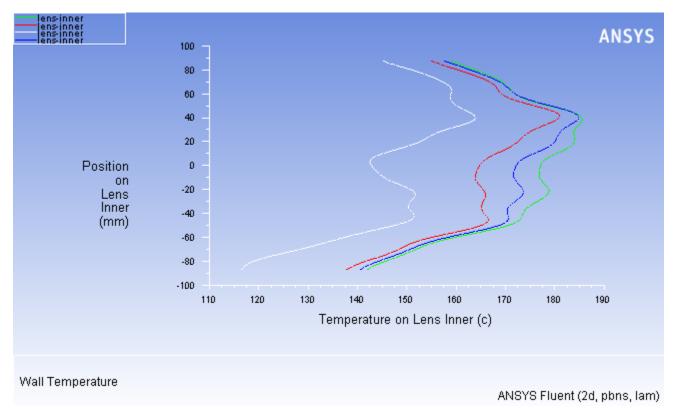
Make sure you deselect lens-inner from the Surfaces list so that there is no duplicated plot.

c. Click the Curves... button to open Curves - Solution XY Plot dialog box.

Curves -	- Solution XY Plot	—	
Curve #	Line Style Pattern Color foreground Weight 1	Marker Style Symbol Color foreground Size 0.3	
Apply Close Help			

- i. Select the line pattern as shown in the Curves Solution XY Plot dialog box.
- ii. Select the symbol pattern as shown in the **Curves Solution XY Plot** dialog box.
- iii. Click **Apply** to save the settings for curve zero.
- iv. Set **Curve #** to 1.
- v. Follow the above instructions for curves 2, 3, and 4.
- vi. Click Apply and close the Curves Solution XY Plot dialog box.
- d. Click Plot (Figure 8.8: Temperature Profile (p. 392)).
- e. Close the Solution XY Plot dialog box.

Figure 8.8: Temperature Profile



8.4.11. Iterate for Higher Divisions

1. Retain the default division as a base for comparison.

```
\clubsuit Models \rightarrow \blacksquare Radiation \rightarrow Edit...
```

Radiation Model			
Model	Iteration Parameter	s	
 Off Rosseland P1 		Flow Iteration	ons per Radiation Iteration 1
Discrete Transfer (DTRM)	Angular Discretizatio	on	Non-Gray Model
 Surface to Surface (S2S) Discrete Ordinates (DO) 	Theta Divisions	2	Number of Bands 0
DO/Energy Coupling	Phi Divisions 2	2	
	Theta Pixels	3	
	Phi Pixels	3	
OK Cancel Help			

a. Retain both Theta Divisions and Phi Divisions as 2.

- b. Enter a value of 3 for Theta Pixels and Phi Pixels.
- c. Click OK to close the Radiation Model dialog box.

This creates a baseline giving better solution efficiency.

2. Request 1500 more iterations.

CRun Calculation

The solution will converge in approximately 500 iterations.

3. Save the case and data files (do_2x2_3x3_div.cas.gz and do_2x2_3x3_div.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

4. Display temperature profiles for the **lens-inner**.

$\textcircled{Plots} \rightarrow \overleftarrow{E} XY Plot \rightarrow Set Up...$

- a. Select all the files from the File Data selection list.
- b. Click Free Data to remove the files from the list.
- c. Retain the settings for Y axis Function and X axis Function.
- d. Select lens-inner from the Surfaces selection list.
- e. Click Plot.
- f. Enable Write to File and click the Write... button to open the Select File dialog box.
 - i. Enter do_2x2_3x3_div.xy for XY File and close the Select File dialog box.
- 5. Repeat the procedure for 3 Theta Divisions and Phi Divisions.
 - a. Save the file as do_3x3_3x3_div.xy.
- 6. Save the case and data files (do_3x3_3x3_div.cas.gz and do_3x3_3x3_div.dat.gz).

File \rightarrow Write \rightarrow Case & Data...

- 7. Repeat the procedure for 5 Theta Divisions and Phi Divisions.
 - a. Save the file as do_5x5_3x3_div.xy.
- 8. Read in all the files for **Theta Divisions** and **Phi Divisions** of 2, 3, and 5 and display temperature profiles. *Make sure you deselect lens-inner from the Surfaces list so that no plots are duplicated.*

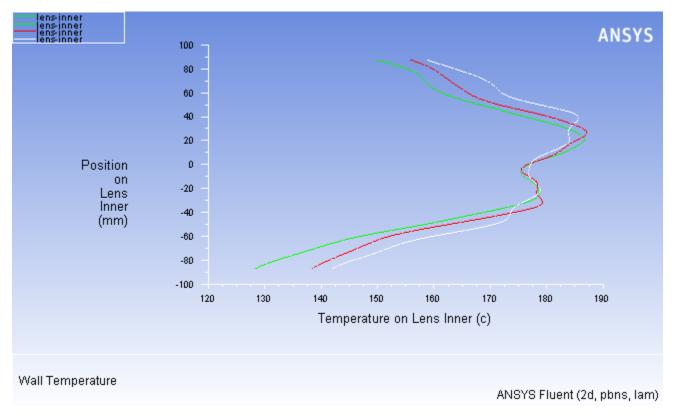


Figure 8.9: Temperature Profiles for Various Theta Divisions

9. Save the case and data files (do_5x5_3x3_div.cas.gz and do_5x5_3x3_div.dat.gz).

File \rightarrow Write \rightarrow Case & Data...

10. Compute the total heat transfer rate.

 $\mathbf{O} \mathsf{Reports} \to \mathbf{Fluxes} \to \mathsf{Set} \mathsf{Up...}$

Iux Reports		—
Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types axis exhaust-fan fan inlet-vent Save Output Parameter	Boundaries	Results -33.67927932739258 33.67926406860352 596.8300170898438 -47.63700866699219 -549.2214965820313 549.216552734375 -549.2379760742188 0 ✓ III Net Results (w) -0.04992676
Compute Write	Close Hel	P

- a. Select Total Heat Transfer Rate in the Options group box.
- b. Select all zones from the **Boundaries** selection list.
- c. Click **Compute**.

Note

The net heat load is -0.0499 W, which equates to an imbalance of approximately 0.008% when compared against the heat load of the bulb.

11. Compute the radiation heat transfer rate.

 $\textcircled{P} Reports \rightarrow \fbox{Fluxes} \rightarrow Set Up...$

Flux Reports		×
Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types axis exhaust-fan fan inlet-vent Y Boundary Name Pattern Save Output Parameter Compute Write.	Boundaries ir-interior baffle baffle-shadow bulb-outer housing lens-inner lens-inner-shadow lens-interior lens-outer lens-sides reflector Close Help	-41.79331207275391 17.37430191040039 522.9666137695313 -18.78656578063965 -464.8591613769531 464.8591613769531 -310.2204895019531 -1.051566123962402 -14.92804050445557 ✓ III Net Results (w) 153.5609

- a. Select Radiation Heat Transfer Rate in the Options group box.
- b. Retain the selection of all boundary zones from the Boundaries selection list.
- c. Click Compute and close the Flux Reports dialog box.

Note

The net heat load is approximately 154 W.

12. Compute the radiation heat transfer rate incident on the surfaces.



Surface Integrals	×
Report Type	Field Variable
Integral 👻	Wall Fluxes 🔻
Surface Types 🗵 🔳 🚍	Surface Incident Radiation 🗸
axis	Surfaces 🔋 🗏 🗏
exhaust-fan	air-interior
fan 👻	baffle baffle-shadow
Surface Name Pattern	bulb-outer
	housing 🗧
Match	lens
	lens-inner lens-inner-shadow
	ens-inter-shadow
	lens-outer
	lens-sides 🔻
	Integral (w/m2)(m2)
Save Output Parameter	4703.993
Compute Write.	Close Help

- a. Select Integral from the Report Type drop-down list.
- b. Select Wall Fluxes... and Surface Incident Radiation from the Field Variable drop-down lists.
- c. Select all surfaces except **air-interior** and **lens-interior** from the **Surfaces** selection list.
- d. Click **Compute**.
- 13. Compute the reflected radiation flux.

 $\clubsuit Reports \rightarrow \overleftarrow{E} Surface Integrals \rightarrow Set Up...$

Surface Integrals		X
Report Type	Field Variable	
Integral 🗸	Wall Fluxes	•
Surface Types	Reflected Radiation Flux	•
axis clip-surf	Surfaces	
exhaust-fan	air-interior	*
fan 👻	baffle	
L . .	baffle-shadow	
Surface Name Pattern	bulb-outer	
Match	housing lens	E
	lens-inner	
	lens-inner-shadow	
	lens-interior	
	lens-outer	-
	lens-sides	
	Integral (w/m2)(m2)	
Save Output Parameter	2881.762	
Compute Write.	. Close Help	

a. Retain the selection of Integral from the Report Type drop-down list.

b. Select Wall Fluxes... and Reflected Radiation Flux from the Field Variable drop-down lists.

- c. Select all surfaces except air-interior and lens-interior from the Surfaces selection list.
- d. Click Compute.

Reflected radiation flux values are printed in the console for all the zones. The zone **baffle** is facing the filament and its shadow (**baffle-shadow**) is facing the lens. There is much more reflection on the filament side than on the lens side, as expected.

lens-inner is facing the fluid and **lens-inner-shadow** is facing the lens. Due to different refractive indexes and non-zero absorption coefficient on the lens, there is some reflection at the interface. Reflection on **lens-inner-shadow** is the reflected energy of the incident radiation from the lens side. Reflection on **lensinner** is the reflected energy of the incident radiation from the fluid side.

14. Compute the transmitted radiation flux.

 $\clubsuit Reports \rightarrow \overleftarrow{E} Surface Integrals \rightarrow Set Up...$

Surface Integrals	×
Report Type	Field Variable
Integral 🗸	Wall Fluxes 🗸
Surface Types	Transmitted Radiation Flux 🗸
axis	Surfaces 🗦 🗏 🚍
exhaust-fan	air-interior
fan 👻	baffle baffle-shadow
Surface Name Pattern	bulb-outer
	housing E
Match	lens
	lens-inner lens-inner-shadow
	lens-interior
	lens-outer
	Integral (w/m2)(m2)
	1445.696
Save Output Parameter	1113.030
Compute Write.	. Close Help

- a. Retain the selection of Integral from the Report Type drop-down list.
- b. Select Wall Fluxes... and Transmitted Radiation Flux from the Field Variable drop-down lists.
- c. Ensure that all surfaces are selected except **air-interior** and **lens-interior** from the **Surfaces** selection list.

d. Click Compute.

Transmitted radiation flux values are printed in the console for all the zones. All surfaces are opaque except lens. Zero transmission for all surfaces indicate that they are opaque.

15. Compute the absorbed radiation flux.

 $\clubsuit Reports \rightarrow \blacksquare Surface Integrals \rightarrow Set Up...$

Surface Integrals		×
Report Type	Field Variable	
Integral 🗸 🗸	Wall Fluxes	•
Surface Types 🗵 🗏 🚍	Absorbed Radiation Flux	•
axis dip-surf	Surfaces	
exhaust-fan	air-interior	
fan 👻	baffle	
Surface Name Pattern	baffle-shadow bulb-outer	
Surface Name Pattern	housing	Ξ
Match	lens	-
	lens-inner	
	lens-inner-shadow	
	lens-interior	
	lens-outer lens-sides	Ŧ
	Integral (w/m2)(m2)	
Save Output Parameter	376.5431	
Compute Write	Close Help	

- a. Retain the selection of Integral from the Report Type drop-down list.
- b. Select Wall Fluxes... and Absorbed Radiation Flux from the Field Variable drop-down lists.
- c. Ensure that all surfaces are selected except **air-interior** and **lens-interior** from the **Surfaces** selection list.
- d. Click Compute.
- e. Close the Surface Integrals dialog box.

Absorption will only occur on opaque surface with a non-zero internal emissivity adjacent to participating cell zones. Note that absorption will not occur on a semi-transparent wall (irrespective of the setting for internal emissivity). In semi-transparent media, absorption and emission will only occur as a volumetric effect in the participating media with non-zero absorption coefficients.

8.4.12. Make the Reflector Completely Diffuse

- 1. Read in the case and data files (do_3x3_3x3_div.cas.gz and do_3x3_3x3_div.dat.gz).
- 2. Increase the diffuse fraction for **reflector**.

 $\mathbf{O} \mathbf{Boundary} \mathbf{Conditions} \rightarrow \mathbf{E} \mathbf{reflector} \rightarrow \mathbf{Edit...}$

💶 Wall	
Zone Name	
reflector	
Adjacent Cell Zone	
fluid	-
Momentum Thermal Radiation Species DPM Multiphase	UDS Wall Film
BC Type	
opaque 👻	
Diffuse Fraction	
OK Can	cel Help

- a. Click the **Radiation** tab and enter 1 for **Diffuse Fraction**.
- b. Click **OK** to close the **Wall** dialog box.
- 3. Request another 1500 iterations.

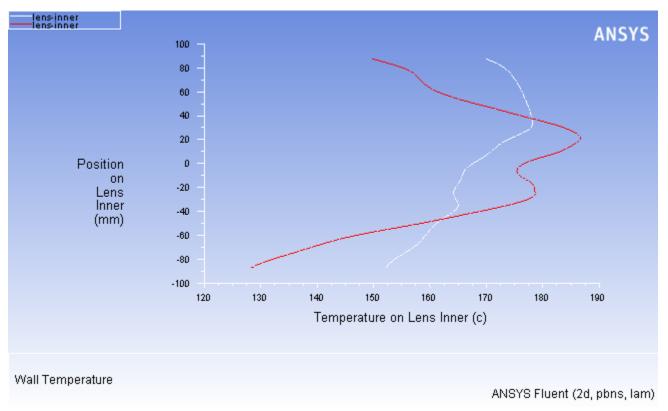
CRun Calculation

The solution will converge in approximately 700 additional iterations.

4. Plot the temperature profiles with the increased diffuse fraction for the **reflector**.

- a. Save the file as do_3x3_3x3_div_df=1.xy.
- b. Save the case and data files as do_3x3_3x3_div_df1.cas.gz and do_3x3_3x3_div_df1.dat.gz.





8.4.13. Change the Boundary Type of Baffle

- 1. Read in the case and data files (do_3x3_3x3_div.cas.gz and do_3x3_3x3_div.dat.gz).
- 2. Change the boundary type of **baffle** to **interior**.

```
↔ Boundary Conditions \rightarrow \stackrel{\frown}{=} baffle
```

a. Select interior from the Type drop-down list.

A **Question** dialog box will open, asking if you want to change **Type** of **baffle** to **interior**.

Question		×
?	OK to change baffle's type from wall to interior?	
	Yes No	

b. Click **Yes** in the **Question** dialog box.

Interior		×
Zone Name baffle		
	OK Cancel Help	

- c. Click **OK** in the **Interior** dialog box.
- 3. Reduce the under-relaxation factors.

Colution Controls

- a. Enter 0.5 for Pressure.
- b. Enter 0.3 for Momentum.
- 4. Request another 2000 iterations.

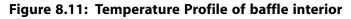
CRun Calculation

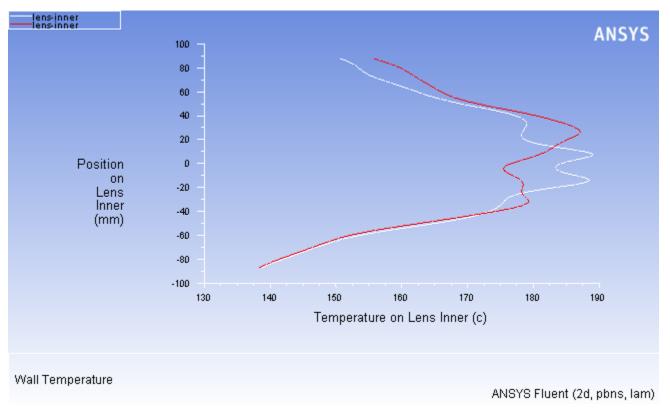
The solution will converge in approximately 1750 additional iterations.

5. Plot the temperature profile for lens-inner based on the modified baffle.

$$\mathbf{O} \mathsf{Plots} \to \mathbf{F} \mathsf{XY} \mathsf{Plot} \to \mathsf{Set} \mathsf{Up...}$$

- a. Save the file as do_3x3_3x3_div_baf_int.xy.
- b. Save the case and data files as do_3x3_3x3_div_int.cas.gz and do_3x3_3x3_div_int.dat.gz.





8.5. Summary

This tutorial demonstrated the modeling of radiation using the discrete ordinates (DO) radiation model in ANSYS Fluent. In this tutorial, you learned the use of angular discretization and pixelation available in the discrete ordinates radiation model and solved for different values of **Pixels** and **Divisions**. You studied the change in behavior for higher absorption coefficient. Changes in internal emissivity, refractive index, and diffuse fraction are illustrated with the temperature profile plots.

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. 123).

Chapter 9: Using a Non-Conformal Mesh

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

9.1. Introduction

Film cooling is a process that is used to protect turbine vanes in a gas turbine engine from exposure to hot combustion gases. This tutorial illustrates how to set up and solve a film cooling problem using a non-conformal mesh. The system that is modeled consists of three parts: a duct, a hole array, and a plenum. The duct is modeled using a hexahedral mesh, and the plenum and hole regions are modeled using a tetrahedral mesh. These two meshes are merged together to form a "hybrid" mesh, with a non-conformal interface boundary between them.

Due to the symmetry of the hole array, only a portion of the geometry is modeled in ANSYS Fluent, with symmetry applied to the outer boundaries. The duct contains a high-velocity fluid in streamwise flow (Figure 9.1: Schematic of the Problem (p. 406)). An array of holes intersects the duct at an inclined angle, and a cooler fluid is injected into the holes from a plenum. The coolant that moves through the holes acts to cool the surface of the duct, downstream of the injection. Both fluids are air, and the flow is classified as turbulent. The velocity and temperature of the streamwise and cross-flow fluids are known, and ANSYS Fluent is used to predict the flow and temperature fields that result from convective heat transfer.

This tutorial demonstrates how to do the following:

- Merge hexahedral and tetrahedral meshes to form a hybrid mesh.
- · Create a non-conformal mesh interface.
- Model heat transfer across a non-conformal interface with specified temperature and velocity boundary conditions.
- Calculate a solution using the pressure-based solver.
- Plot temperature profiles on specified iso-surfaces.

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. 123)

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

9.3. Problem Description

This problem considers a model of a 3D section of a film cooling test rig. A schematic of the problem is shown in Figure 9.1: Schematic of the Problem (p. 406). The problem consists of a duct, 49 inches long, with cross-sectional dimensions of 0.75 inches \times 5 inches. An array of uniformly-spaced holes is located at the bottom of the duct. Each hole has a diameter of 0.5 inches, is inclined at 35 degrees, and is spaced 1.5 inches apart laterally. Cooler injected air enters the system through the plenum having cross-sectional dimensions of 3.3 inches \times 1.25 inches.

Only a portion of the domain must be modeled because of the symmetry of the geometry. The bulk temperature of the streamwise air (T_{∞}) is 450 K, and the velocity of the air stream is 20 m/s. The bottom wall of the duct that intersects the hole array is assumed to be a completely insulated (adiabatic) wall. The secondary (injected) air enters the plenum at a uniform velocity of 0.4559 m/s. The temperature of the injected air (T_{inject}) is 300 K. The properties of air that are used in the model are also mentioned in Figure 9.1: Schematic of the Problem (p. 406).

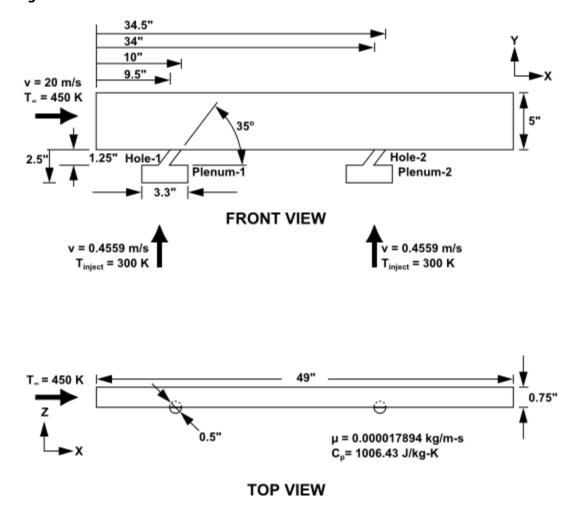


Figure 9.1: Schematic of the Problem

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. Operating Conditions
9.4.8. Boundary Conditions
9.4.9. Mesh Interfaces
9.4.10. Solution
9.4.11. Postprocessing

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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip non_conformal_mesh_R150.zip to your working folder.

The input files film_hex.msh and film_tet.msh can be found in the non_conformal_mesh 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 Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Run in single precision (disable **Double Precision**).
- 11. Ensure you are running in Serial under Processing Options.

9.4.2. Mesh

1. Read the hex mesh file **film_hex.msh**.

```
File \rightarrow Read \rightarrow Mesh...
```

2. Append the tet mesh file **film_tet.msh**.

```
\textbf{Mesh} \rightarrow \textbf{Zone} \rightarrow \textbf{Append Case File...}
```

The Append Case File... functionality allows you to combine two mesh files into one single mesh file.

3. Check the mesh.

General → 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.

4. Scale the mesh and change the unit of length to inches.

$\textcircled{} General \rightarrow Scale...$

Scale Mesh		—
Domain Extents		Scaling
Xmin (m)9.5	Xmax (m) 39.5	 Convert Units Specify Scaling Factors
Ymin (m) -2,5	Ymax (m) 5.000001	Mesh Was Created In
Zmin (m)	Zmax (m) 0.75	Scaling Factors
View Length Unit In		× 0.0254
m •		Y 0.0254
		Z 0.0254
		Scale Unscale
	Close Help	

- a. Ensure that **Convert Units** is selected in the **Scaling** group box.
- b. Select **in** from the **Mesh Was Created In** drop-down list 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.

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. Close the Scale Mesh dialog box.
- 5. Check the mesh.

ĢGeneral → 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.

6. Display an outline of the 3D mesh.

$\bigcirc General \rightarrow Display...$

- a. Retain the default selections in the Surfaces list.
- b. Click Display.
- c. Close the Mesh Display dialog box.
- 7. Manipulate the mesh display to obtain a front view as shown in Figure 9.2: Hybrid Mesh for Film Cooling Problem (p. 410).

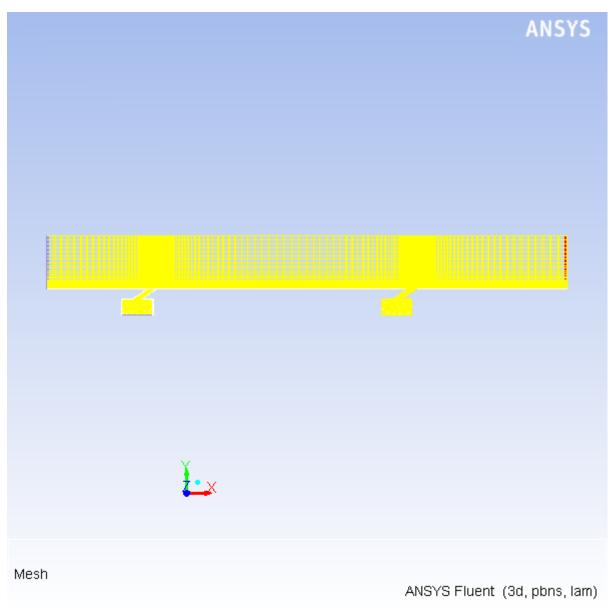
Graphics and Animations → Views...

Q Views		—
Views back bottom front isometric left right top Save Name front	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes = symmetry-tet1 symmetry-tet2 symmetry-5 symmetry-1 symmetry-7 Define Plane Periodic Repeats Define
Apply Camer	ra Close	Help

a. Select front in the Views list.

- b. Click Apply.
- c. Close the Views dialog box.

Figure 9.2: Hybrid Mesh for Film Cooling Problem



8. Zoom in using the middle mouse button to view the hole and plenum regions (Figure 9.3: Hybrid Mesh (Zoomed-In View) (p. 411)).

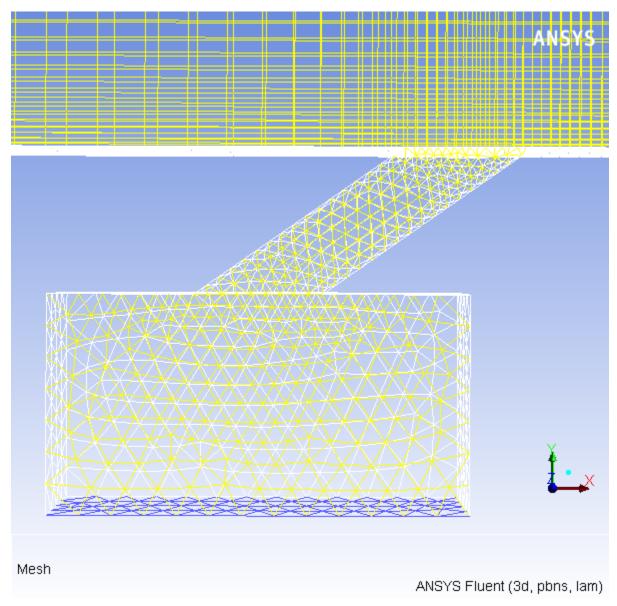


Figure 9.3: Hybrid Mesh (Zoomed-In View)

In Figure 9.3: Hybrid Mesh (Zoomed-In View) (p. 411), you can see the quadrilateral faces of the hexahedral cells that are used to model the duct region and the triangular faces of the tetrahedral cells that are used to model the plenum and hole regions, resulting in a hybrid mesh.

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.

9.4.3. General Settings

1. Retain the default solver settings.

General

General	
Mesh	
Scale Display	Check Report Quality
Solver	
Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative
Time ◉ Steady ⓒ Transient	
Gravity	Units
Help	

9.4.4. Models

1. Enable heat transfer by enabling the energy equation.

Energy	×
Energy	1
Energy Equation	
OK Cancel	Help

- a. Click **OK** to close the **Energy** dialog box.
- 2. Enable the standard k- ε turbulence model.

a. Select k-epsilon (2eqn) in the Model list.

The **Viscous Model** dialog box will expand to show the additional input options for the k- ε model.

b. 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 Setting Up the k- ε Model in the User's Guide.

- c. Retain the default settings for the remaining parameters.
- d. Click OK to close the Viscous Model dialog box.

9.4.5. Materials

- 1. Define the material properties.
 - $\mathbf{O}_{\mathsf{Materials}} \rightarrow \mathbf{Fluid} \rightarrow \mathsf{Create/Edit...}$

Create/Edit Materials		—
Name	Material Type	Order Materials by
Chemical Formula	FLUENT Fluid Materials	Chemical Formula
	air	 FLUENT Database
	Mixture	User-Defined Database
	none	Ŧ
Properties		
Density (kg/m3)	incompressible-ideal-gas Edit	
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 1.7894e-05	
	Change/Create Delete Close Help	

- a. Retain the selection of air from the Fluent Fluid Materials drop-down list.
- b. Select incompressible-ideal-gas law from the Density drop-down list.

The incompressible ideal gas law is used when pressure variations are small but temperature variations are large. The incompressible ideal gas option for density treats the fluid density as a function of temperature only. If the above condition is satisfied, the incompressible ideal gas law generally gives better convergence compared to the ideal gas law, without sacrificing accuracy.

- c. Retain the default values for all other properties.
- d. Click Change/Create and close the Create/Edit Materials dialog box.

9.4.6. Cell Zone Conditions

1. Set the conditions for the fluid in the duct (fluid-9).

```
\textcircled{Cell Zone Conditions} \rightarrow \overleftarrow{\sqsubseteq} fluid-9 \rightarrow Edit...
```

💶 Fluid							- X-	
Zone Name								
fluid-duct								
Material Na	Material Name air 🗸 Edit							
	Frame Motion Laminar Zone Source Terms							
Mesh Motion Fixed Values								
	Porous Zone Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Multiphase							
	erreine Thear	Houding Porods zone Enibed	Jueur	LES Reaction	Source remis	T tixed values [_	
Rotatio	Rotation-Axis Origin			Rotation-Axis Direction				
X (in)	0	constant 👻	X	0	constant	•		
Y (in)	0	constant 👻	Y	0	constant	•		
Z (in)	ļ			1				
200	0	constant 👻	-	1	constant	•		
			_				Ţ	
OK Cancel Help								

- a. Change the **Zone Name** from fluid-9 to fluid-duct.
- b. Retain the default selection of air from the Material Name drop-down list.
- c. Click **OK** to close the **Fluid** dialog box.
- 2. Set the conditions for the fluid in the first plenum and hole (**fluid-8**).

$\bigcirc Cell Zone Conditions \rightarrow \blacksquare fluid-8 \rightarrow Edit...$

- a. Change the **Zone Name** from fluid-8 to fluid-plenum1.
- b. Retain the default selection of air from the Material Name drop-down list.
- c. Click **OK** to close the **Fluid** dialog box.
- 3. Set the conditions for the fluid in the second plenum and hole (**fluid-9.1**).

♦ Cell Zone Conditions $\rightarrow \stackrel{\frown}{=}$ fluid-9.1 \rightarrow Edit...

- a. Change the **Zone Name** from fluid-9.1 to fluid-plenum2.
- b. Retain the default selection of air from the Material Name drop-down list.
- c. Click **OK** to close the **Fluid** dialog box.

9.4.7. Operating Conditions

Generating Conditions → Operating Conditions...

1. Retain the default operating conditions.

Operating Conditions	×					
Pressure	Gravity					
Operating Pressure (pascal)	Gravity					
101325 P						
Reference Pressure Location						
X (in) 0						
Y (in) 0						
Z (in) 0						
OK Cancel Help						

2. Click OK to close the Operating Conditions dialog box.

For the **incompressible-ideal-gas law** selected here for air, the constant pressure used for the density calculation is the **Operating Pressure** specified in this dialog box. So, make sure that the **Operating Pressure** is close to the mean pressure of the domain.

9.4.8. Boundary Conditions

1. Set the boundary conditions for the streamwise flow inlet (velocity-inlet-1).

 $\textcircled{P} Boundary \ Conditions \rightarrow \fbox{velocity-inlet-1} \rightarrow Edit...$

Velocity Inlet			×
Zone Name velocity-inlet-duct			
		L.	
Momentum Thermal Radiation Specie	s DPM Multiphase U	DS	
Velocity Specification Method	Magnitude, Normal to Bour	ndary	-
Reference Frame	Absolute		•
Velocity Magnitude (m/s)	20	constant	•
Supersonic/Initial Gauge Pressure (pascal)	0	constant	•
Turbulence			
Specification Method	Intensity and Hydraulic Diam	eter	•
	Turbulent Intensity (%	6) 1	e
	Hydraulic Diameter (ii	n) 5	e
	Cancel Help		

- a. Change the **Zone Name** from velocity-inlet-1 to velocity-inlet-duct.
- b. Enter 20 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 1% and 5 in for the **Turbulent Intensity** and the **Hydraulic Diameter**, respectively.
- e. Click the **Thermal** tab and enter 450 K for the **Temperature**.
- f. Click **OK** to close the **Velocity Inlet** dialog box.
- 2. Set the boundary conditions for the first injected stream inlet (velocity-inlet-5).

♀ Boundary Conditions → \blacksquare velocity-inlet-5 → Edit...

Velocity Inlet		×
Zone Name		
velocity-inlet-plenum1		
Momentum Thermal Radiation Species	DPM Multiphase U	os
Velocity Specification Method	Magnitude, Normal to Boun	dary 🔹
Reference Frame	Absolute	•
Velocity Magnitude (m/s)	0.4559	constant 💌
Supersonic/Initial Gauge Pressure (pascal)	0	constant 💌
Turbulence		
Specification Method [ntensity and Viscosity Ratio	•
	Turbulent Intensity (%	6) 1 P
	Turbulent Viscosity Rati	10 E
ОК	Cancel Help	

- a. Change the **Zone Name** from velocity-inlet-5 to velocity-inlet-plenum1.
- b. Enter 0.4559 m/s for the Velocity Magnitude.
- c. Retain **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.
- d. Enter 1% for **Turbulent Intensity** and retain the default setting of 10 for **Turbulent Viscosity Ratio**.
- e. Click the **Thermal** tab and retain the setting of 300 K for **Temperature**.
- f. Click **OK** to close the **Velocity Inlet** dialog box.

In the absence of any identifiable length scale for turbulence, the **Intensity and Viscosity Ratio** method should be used.

For more information about setting the boundary conditions for turbulence, see Modeling Turbulence in the User's Guide.

3. Copy the boundary conditions set for the first injected stream inlet.

♣ Boundary Conditions → \blacksquare velocity-inlet-plenum1 → Copy...

Copy Conditions	
From Boundary Zone symmetry-7 symmetry-tet1 symmetry-tet2 velocity-inlet-6 velocity-inlet-duct velocity-inlet-plenum1 wall-1 wall-4 wall-5	To Boundary Zones 🖹 🗐
Сору	Close Help

- a. Select velocity-inlet-plenum1 in the From Boundary Zone selection list.
- b. Select velocity-inlet-6 in the To Boundary Zones selection list.
- c. Click Copy.

A **Warning** dialog box will open, asking if you want to copy **velocity-inlet-plenum1** boundary conditions to **(velocity-inlet-6)**. Click **OK**.

d. Close the Copy Conditions dialog box.

Warning

Copying a boundary condition does not create a link from one zone to another. If you want to change the boundary conditions on these zones, you will have to change each one separately.

4. Set the boundary conditions for the second injected stream inlet (velocity-inlet-6).

 $\textcircled{} Boundary \text{ Conditions} \rightarrow \overleftarrow{\sqsubseteq} \text{ velocity-inlet-6} \rightarrow \text{ Edit...}$

💽 Velocity Inlet	x
Zone Name	
velocity-inlet-plenum2	
Momentum Thermal Radiation Species DPM Multiphase UDS	-
Temperature (k) 300 constant	
OK Cancel Help	

- a. Change the **Zone Name** from velocity-inlet-6 to velocity-inlet-plenum2.
- b. Verify that the boundary conditions were copied correctly.
- c. Click **OK** to close the **Velocity Inlet** dialog box.
- 5. Set the boundary conditions for the flow exit (**pressure-outlet-1**).

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{\equiv}$ pressure-outlet-1 \rightarrow Edit...

Pressure Outlet
Zone Name
pressure-outlet-duct
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method Normal to Boundary
Radial Equilibrium Pressure Distribution
Average Pressure Specification
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Viscosity Ratio
Backflow Turbulent Intensity (%) 1
Backflow Turbulent Viscosity Ratio
OK Cancel Help

- a. Change the **Zone Name** from pressure-outlet-1 to pressure-outlet-duct.
- b. Retain the default setting of 0 Pa for Gauge Pressure.
- c. Retain **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.
- d. Enter 1% for **Backflow Turbulent Intensity** and retain the default setting of 10 for **Backflow Turbulent Viscosity Ratio**.
- e. Click the Thermal tab and enter 450 K for Backflow Total Temperature.
- f. Click OK to close the Pressure Outlet dialog box.
- 6. Retain the default boundary conditions for the plenum and hole walls (wall-4 and wall-5).

 $\textcircled{}Boundary \text{ Conditions} \rightarrow \overleftarrow{\boxtimes} \text{ wall-4} \rightarrow \text{ Edit...}$

Nall	
Zone Name	
wall-4	
Adjacent Cell Zone	
fluid-plenum1	
Momentum Thermal Radiation Species DPM Multiphase	UDS Wall Film
Wall Motion Motion	
Stationary Wall Moving Wall	
Shear Condition	
 No Slip Specified Shear Specularity Coefficient Marangoni Stress 	
Wall Roughness	
Roughness Height (in) 0 constant	·
Roughness Constant 0.5 constant	Y
OK Can	el Help

7. Verify that the symmetry planes are set to the correct type in the **Boundary Conditions** task page.

Boundary Conditions

Boundary Co	nditions	
Zone		
int_interior-1		
interior-3		
interior-4		
pressure-outlet-o	Juct	-1
symmetry-1		
symmetry-5 symmetry-7		
symmetry-tet1		
symmetry-tet2		
velocity-inlet-duc	t	=
velocity-inlet-pler		
velocity-inlet-pler		
wall-1		
wall-4		
wall-5		
wall-7		
wall-8		-
1		
Phase	Type ID	
mixture	✓ symmetry ▼ 6	
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh	Periodic Conditions	
Highlight Zone		
Help		

- a. Select **symmetry-1** in the **Zone** list.
- b. Ensure that **symmetry** is selected from the **Type** drop-down list.
- c. Similarly, verify that the zones **symmetry-5**, **symmetry-7**, **symmetry-tet1**, and **symmetry-tet2** are set to the correct type.
- 8. Define the zones on the non-conformal boundary as interface zones by changing the **Type** for **wall-1**, **wall-7**, and **wall-8** to **interface**.

The non-conformal mesh interface contains three boundary zones: **wall-1**, **wall-7**, and **wall-8**. **wall-1** is the bottom surface of the duct, **wall-7** and **wall-8** represent the holes through which the cool air is injected from the plenum (Figure 9.4: Mesh for the wall-1 and wall-7 Boundaries (p. 425)). These boundaries were defined as walls in the original mesh files (film_hex.msh and film_tet.msh) and must be redefined as **interface** boundary types.

a. Open the Mesh Display dialog box.

 \bigcirc General \rightarrow Display...

💶 Mesh Displa	у		×
Options Nodes Edges Faces	Edge Type	Surfaces +velocity-inlet- [3,0] -wall- [5,3]	
0	Feature Angle	4 5 7 8	E
Outline Inter	Match	New Surface Surface Types axis clip-surf exhaust-fan fan	
	Display Colo	rs Close Help	

- i. Deselect all surfaces by clicking \equiv to the far right of **Surfaces**.
- ii. Collapse the list of surfaces by clicking 📃
- iii. Expand the wall branch by Ctrl + left-clicking +wall- [5,0] in the Surfaces group box.
- iv. Select wall-1, wall-7, and wall-8 from the Surfaces selection list.

Use the scrollbar to access the surfaces that are not initially visible.

- v. Click **Display** and close the **Mesh Display** dialog box.
- b. Display the bottom view.

• Graphics and Animations \rightarrow Views...

- i. Select **bottom** in the **Views** list and click **Apply**.
- ii. Close the **Views** dialog box.

Zoom in using the middle mouse button. Figure 9.4: Mesh for the wall-1 and wall-7 Boundaries (p. 425) shows the mesh for the **wall-1** and **wall-7** boundaries (that is, hole-1). Similarly, you can zoom in to see the mesh for the **wall-1** and **wall-8** boundaries (that is, hole-2).

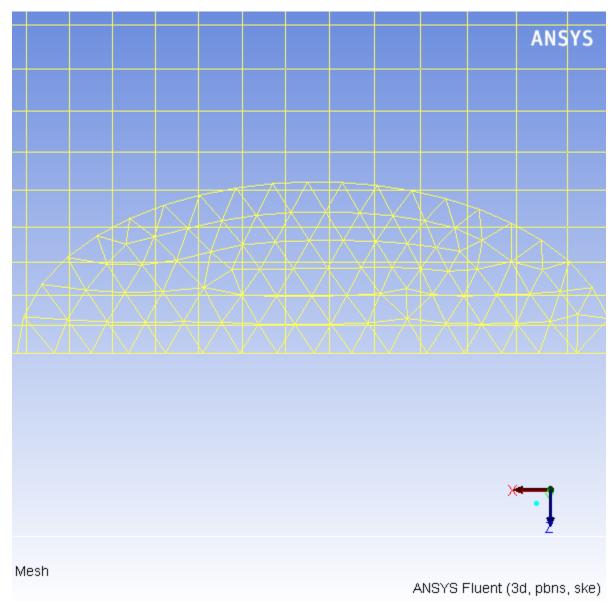


Figure 9.4: Mesh for the wall-1 and wall-7 Boundaries

c. Select wall-1 in the Zone list and select interface as the new Type.



Boundary Conditions	
Zone	
int_interior-1 interior-3 interior-4 pressure-outlet-duct symmetry-1 symmetry-5 symmetry-7 symmetry-tet1 symmetry-tet2 velocity-inlet-duct velocity-inlet-plenum1 velocity-inlet-plenum2 wall-4	•
wall-4 wall-5 wall-7 wall-8	•
mixture v interface v 4	
Edit Copy Profiles Parameters Operating Conditions Display Mesh Periodic Conditions Highlight Zone	
Help	

A **Question** dialog box will open, asking if it is **OK** to change the type of **wall-1** from **wall** to **interface**. Click **Yes** in the **Question** dialog box.

The *Interface* dialog box will open and give the default name for the newly-created interface zone.

Interface		×
Zone Name		
interface-duct		
	OK Cancel Help	

- i. Change the Zone Name to interface-duct.
- ii. Click **OK** to close the **Interface** dialog box.
- d. Similarly, convert wall-7 and wall-8 to interface boundary zones, specifying interface-hole1 and interface-hole2 for Zone Name, respectively.

9.4.9. Mesh Interfaces

In this step, you will create a non-conformal mesh interface between the hexahedral and tetrahedral meshes.

Mesh Interfaces –	→ Create/Edit
-------------------	---------------

			×
Mesh Interface	Interface Zone 1	Interface Zone 2	
junction	interface-hole1, interface-hole2	interface-duct	
junction	interface-duct	interface-duct	
	interface-hole1	interface-hole 1	
	interface-hole2	interface-hole2	
Interface Options	Boundary Zone 1	Interface Wall Zone 1	
Periodic Boundary Condition	wall-25 , wall-24		
Periodic Repeats	Boundary Zone 2	Interface Wall Zone 2	
Coupled Wall Matching	wall-26		
		Interface Interior Zone	
		interior-23, interior-22	
Periodic Boundary Condition			
Type Offset			
Translational X (in) O	Y (in) 0 Z (in	n) 0	
✓ Auto Compute Offset			

1. Select interface-hole1 and interface-hole2 in the Interface Zone 1 selection list.

Warning	
When one interface zone is smaller than the other, choose the smaller zone as Inte face Zone 1.	؛r -

- 2. Select interface-duct from the Interface Zone 2 selection list.
- 3. Enter junction for Mesh Interface.
- 4. Click Create.

In the process of creating the mesh interface, ANSYS Fluent will create three new wall boundary zones: **wall-24**, **wall-25**, and **wall-26**.

 wall-24 and wall-25 are the non-overlapping regions of the interface-hole1 and interface-hole2 zones that result from the intersection of the interface-hole1, interface-hole2, and interface-duct boundary zones. They are listed under Boundary Zone 1 in the Create/Edit Mesh Interfaces dialog box. These wall boundaries are empty, since interface-hole1 and interface-hole2 are completely contained within the interface-duct boundary. • **wall-26** is the non-overlapping region of the **interface-duct** zone that results from the intersection of the three interface zones, and is listed under **Boundary Zone 2** in the **Create/Edit Mesh Interfaces** dialog box.

You will **not** be able to display these walls.

Warning

You need to set boundary conditions for **wall-26** (since it is not empty). In this case, the default settings are used.

5. Close the Create/Edit Mesh Interfaces dialog box.

9.4.10. Solution

1. Set the solution parameters.

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	A
Green-Gauss Cell Based 🔹	
Pressure	
Second Order 🔹	Ξ
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	
High Order Term Relaxation Options	
Default	
(
Help	

a. Select Coupled from the Scheme drop-down list.

- b. Select Second Order Upwind from the Turbulent Kinetic Energy and Turbulent Dissipation Rate drop-down lists in the Spatial Discretization group box.
- 2. Enable the plotting of residuals.

 $\textcircled{} Monitors \rightarrow \overleftarrow{\equiv} Residuals \rightarrow Edit...$

💶 Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monitor C	Check Convergence	e Absolute Criteria	*
V Plot	continuity		V	0.001	
Window 1 Curves Axes	x-velocity	V	v	0.001	- =
Terations to Plot	y-velocity	V	✓	0.001	
1000	z-velocity		\checkmark	0.001	-
	Residual Values			Convergence C	riterion
Iterations to Store	Normalize		Iterations	absolute	-
1000			5		
	Scale				
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** panel.
- 3. Initialize the solution.

Over Solution Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
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.

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.

4. Save the case file (filmcool.cas.gz).

File \rightarrow Write \rightarrow Case...

5. Start the calculation by requesting 250 iterations.

Run Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Number of Iterations	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- a. Enter 250 for the Number of Iterations.
- b. Click Calculate.

The solution converges after approximately 45 iterations.

6. Save the case and data files (filmcool.cas.gz and filmcool.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

Note

If you choose a file name that already exists in the current directory, ANSYS Fluent will prompt you for confirmation to overwrite the file.

9.4.11. Postprocessing

1. Display filled contours of static pressure (Figure 9.5: Contours of Static Pressure (p. 432)).

Contours		×
Options		Contours of
Filled		Pressure 👻
Vode Values Global Range		Static Pressure 🔹
V Auto Range		Min Max
Clip to Range		0
Draw Mesh		Surfaces
		wall-24
Levels Setup	-	wall-25 wall-26
20 🛋 1		wall-4
		wall-5
Surface Name Pattern		New Surface 💌
Ma	atch	Surface Types
		axis
		dip-surf exhaust-fan
		fan 👻
Display		Compute Close Help

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

- a. Enable **Filled** in the **Options** group box.
- b. Ensure that **Pressure...** and **Static Pressure** are selected from the **Contours of** drop-down lists.
- c. Select interface-duct, interface-hole1, interface-hole2, symmetry-1, symmetry-tet1, symmetry-tet2, wall-4, and wall-5 in the Surfaces selection list.

Use the scroll bar to access the surfaces that are not initially visible in the **Contours** dialog box.

d. Click **Display** and close the **Contours** dialog box.



	1.71e+02	ANSYS
	1.60e+02	
	1.48e+02	
	1.36e+02	
	1.25e+02	
	1.13e+02	
	1.02e+02	
	8.99e+01	
	7.83e+01	
	6.67 <mark>e+01</mark>	
	5.51e+01	
	4.34e+01	
	3.18e+01	
	2.02e+01	
	8.57e+00	
	-3.06e+00	
	-1.47e+01	
	-2.63e+01	Y
	-3.79e+01	1
	-4.96e+01	4X
	-6.12e+01	
Cont	ours of Static Pressure (pascal)	
0011		ANSYS Fluent (3d, pbns, ske)

The maximum pressure change (see Figure 9.5: Contours of Static Pressure (p. 432)) is only 232 Pa. Compared to a mean pressure of 1.013e5 Pa, the variation is less than 0.3%, therefore the use of the incompressible ideal gas law is appropriate.

e. Reset the view to the default view if you changed the default display of the mesh.

 $\clubsuit Graphics and Animations \rightarrow Views...$

E Views		×
Views back bottom front isometric left right top Save Name front	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes = symmetry-tet1 symmetry-tet2 symmetry-1 symmetry-7 Define Plane Periodic Repeats Define
Apply Came	ra Close	Help

- i. Click **Default** in the **Actions** group box and close the **Views** dialog box.
- f. Zoom in on the view to display the contours at the holes (Figure 9.6: Contours of Static Pressure at the First Hole (p. 434) and Figure 9.7: Contours of Static Pressure at the Second Hole (p. 435)).

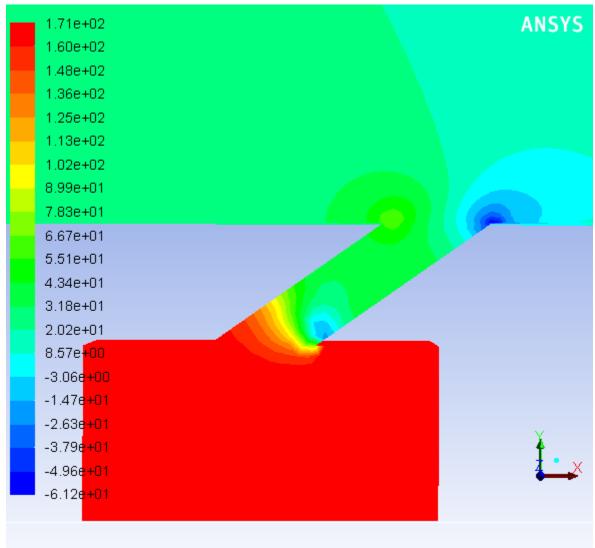


Figure 9.6: Contours of Static Pressure at the First Hole

Contours of Static Pressure (pascal)

ANSYS Fluent (3d, pbns, ske)

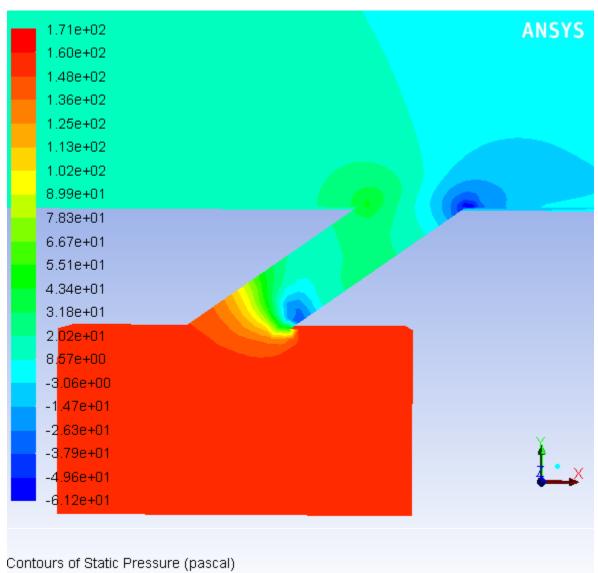


Figure 9.7: Contours of Static Pressure at the Second Hole

ANSYS Fluent (3d, pbns, ske)

Note the high/low pressure zones on the upstream/downstream sides of the coolant hole, where the jet first penetrates the primary flow in the duct.

2. Display filled contours of static temperature (Figure 9.8: Contours of Static Temperature (p. 437) and Figure 9.9: Contours of Static Temperature (Zoomed-In View) (p. 438)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

Contours			×
Options	Contours of		
V Filled	Temperature		•
V Node Values	Static Temperature		-
Global Range Auto Range	Min (k)	Max (k)	
Clip to Range	300	450	
Draw Mesh	Surfaces		
	wall-24		•
Levels Setup	wall-25		
20	wall-26 wall-4		
	wall-5		
Surface Name Pattern	New Surface 🔻		
Match	Surface Types		
	axis		*
	clip-surf		
	exhaust-fan		
	fan		Ŧ
Display Compute Close Help			

- a. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- b. Disable **Auto Range** in the **Options** group box so that you can change the maximum and minimum temperature gradient values to be plotted.
- c. Enter 300 for Min and 450 for Max.
- d. Disable Clip to Range in the Options group box.
- e. Ensure that interface-duct, interface-hole1, interface-hole2, symmetry-1, symmetry-tet1, symmetrytet2, wall-4, and wall-5 are selected from the Surfaces selection list.
- f. Click **Display** and close the **Contours** dialog box.

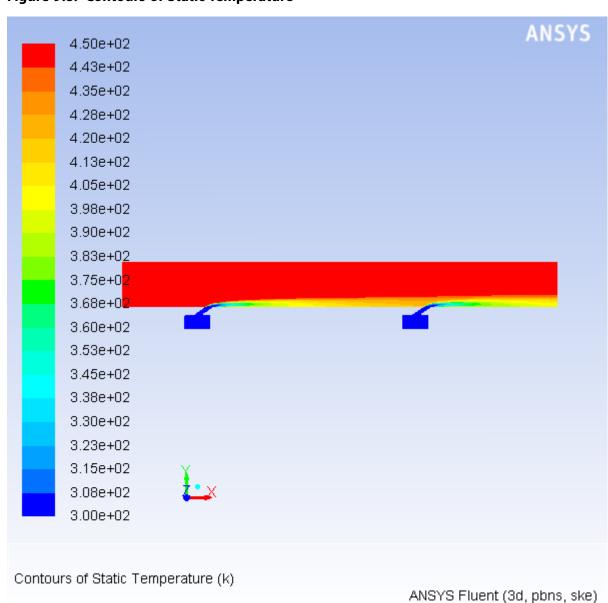


Figure 9.8: Contours of Static Temperature

g. Zoom in on the view to get the display shown in Figure 9.9: Contours of Static Temperature (Zoomed-In View) (p. 438).

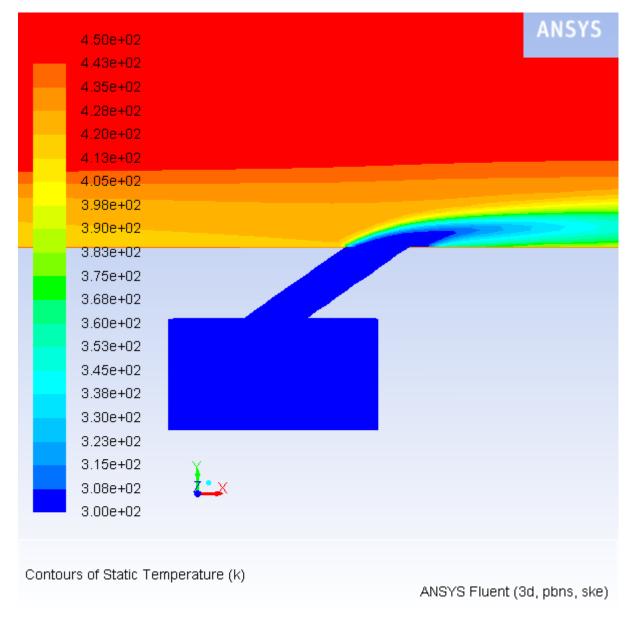


Figure 9.9: Contours of Static Temperature (Zoomed-In View)

Figure 9.8: Contours of Static Temperature (p. 437) and Figure 9.9: Contours of Static Temperature (Zoomed-In View) (p. 438) clearly show how the coolant flow insulates the bottom of the duct from the higher-temperature primary flow.

3. Display the velocity vectors (Figure 9.10: Velocity Vectors (p. 440)).

Graphics and Animations → $\overline{\equiv}$ Vectors → Set Up...

Q Vectors			×
Options	Vectors of		
😺 Global Range	Velocity		•
Auto Range	Color by		
Clip to Range	Velocity		
Draw Mesh	Velocity Magnitude		•
Style	Min (m/s)	Max (m/s)]
arrow	0.004387352	21.52871	
Scale Skip	Surfaces	-	
2 0 🛋	velocity-inlet-plenum2		
Vector Options	wall-24		
	wall-25 wall-26		
Custom Vectors	wall-4		
	wall-5		
Surface Name Pattern			
Match	New Surface 🔻		
	Surface Types		
	axis		
	clip-surf		
	exhaust-fan		
	fan		*
Display	Compute Close	Help	

- a. Ensure that Velocity... and Velocity Magnitude are selected from the Color by drop-down lists.
- b. Ensure that Auto Range is enabled in the Options group box.
- c. Enter 2 for the **Scale**.

This enlarges the displayed vectors, making it easier to view the flow patterns.

d. Select interface-duct, interface-hole1, interface-hole2, symmetry-1, symmetry-tet1, symmetry-tet2, wall-4, and wall-5 from the Surfaces selection list.

Use the scroll bar to access the surfaces that are not initially visible in the dialog box.

- e. Click **Display** and close the **Vectors** dialog box.
- f. Zoom in on the view to get the display shown in Figure 9.10: Velocity Vectors (p. 440).

In Figure 9.10: Velocity Vectors (p. 440), the flow pattern in the vicinity of the coolant hole shows the level of penetration of the coolant jet into the main flow. Note that the velocity field varies smoothly across the non-conformal interface.

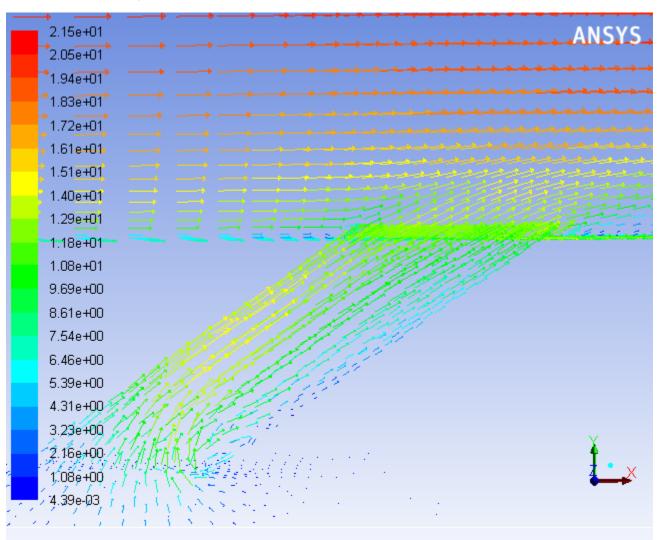


Figure 9.10: Velocity Vectors

Velocity Vectors Colored By Velocity Magnitude (m/s)

ANSYS Fluent (3d, pbns, ske)

4. Create an iso-surface along a horizontal cross-section of the duct, 0.1 inches above the bottom, at y = 0.1 inches.

Surface \rightarrow Iso-Surface...

Iso-Surface		×
Surface of Constant Mesh Y-Coordinate Min Max 0 Iso-Values 0.1	India pichani 1	
New Surface Name y=0. lin Create Compute Manage	fluid-plenum2	

- a. Select Mesh... and Y-Coordinate from the Surface of Constant drop-down lists.
- b. Enter 0.1 for Iso-Values.
- c. Enter y=0.lin for New Surface Name.
- d. Click Create.
- e. Close the Iso-Surface dialog box.
- 5. Create an XY plot of static temperature on the iso-surface created (Figure 9.11: Static Temperature at y=0.1 in (p. 443)).
 - $\textcircled{Plots} \rightarrow \overleftarrow{\blacksquare} XY Plot \rightarrow Set Up...$

Solution XY Plot			×
Options	Plot Direction	Y Axis Function	
Vode Values Vosition on X Axis Position on Y Axis Vrite to File Order Points	X 1	Temperature	•
	YO	Static Temperature	•
		X Axis Function	
	Z O	Direction Vector	
File Data 🗎 🗏 🗏		Surfaces	
]	velocity-inlet-plenum2	*
		wall-24	
		wall-25	
		wall-26	
		wall-4	
		wall-5	E
	Load File	y=0.1in	Ŧ
	Free Data	New Surface 🔻	
	(Ince bata)		
Plot	Axes	Curves Close Help	

- a. Retain the default values in the **Plot Direction** group box.
- b. Select Temperature... and Static Temperature from the Y-Axis Function drop-down lists.
- c. Select **y=0.1in** in the **Surfaces** selection list.

Scroll down using the scroll bar to access **y=0.1in**.

d. Click Plot.

In Figure 9.11: Static Temperature at y=0.1 in (p. 443), you can see how the temperature of the fluid changes as the cool air from the injection holes mixes with the primary flow. The temperature is coolest just downstream of the holes. You can also make a similar plot on the lower wall to examine the wall surface temperature.

e. Close the Solution XY Plot dialog box.

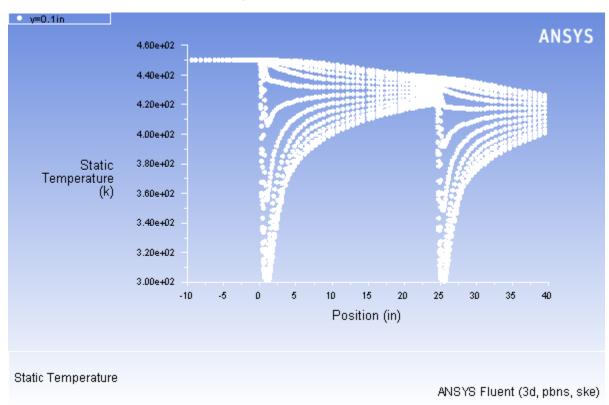


Figure 9.11: Static Temperature at y=0.1 in

9.5. Summary

This tutorial demonstrated how the non-conformal mesh interface capability in ANSYS Fluent can be used to handle hybrid meshes for complex geometries, such as the film cooling hole configuration examined here. One of the principal advantages of this approach is that it allows you to merge existing component meshes together to create a larger, more complex mesh system, without requiring that the different components have the same node locations on their shared boundaries. Thus, you can perform parametric studies by merging the desired meshes, creating the non-conformal interface(s), and solving the model. For example, in the present case, you can do the following:

- Use a different hole/plenum mesh.
- Reposition the existing hole/plenum mesh.
- Add additional hole/plenum meshes to create aligned or staggered multiple-hole arrays.

9.6. 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. 123).

Chapter 10: Modeling Flow Through Porous Media

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 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 used to design 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.

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)

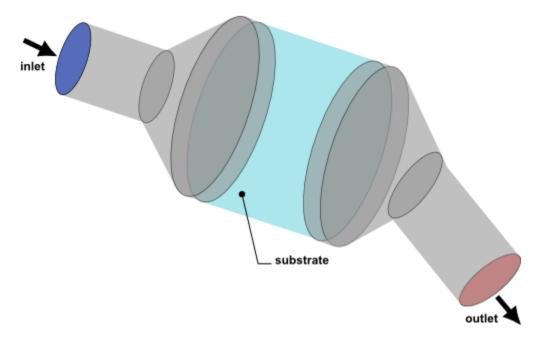
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

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

10.3. Problem Description

The catalytic converter modeled here is shown in Figure 10.1: Catalytic Converter Geometry for Flow Modeling (p. 446). 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.





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.

10.4. Setup and Solution

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

10.4.1. Preparation
10.4.2. Mesh
10.4.3. General Settings
10.4.4. Models
10.4.5. Materials
10.4.6. Cell Zone Conditions
10.4.7. Boundary Conditions
10.4.8. Solution
10.4.9. Postprocessing

10.4.1. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip porous_R150.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 the 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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Enable **Double-Precision**.
- 11. Ensure you are running in **Serial** under **Processing Options**.

10.4.2. Mesh

1. Read the mesh file (catalytic_converter.msh).

 $\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

2. Check the mesh.

$\bigcirc General \rightarrow Check$

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.

Scale Mesh		EX
Domain Extents		Scaling
Xmin (mm) -1.22461e-15	Xmax (mm) 276.7358	 Convert Units Specify Scaling Factors
Ymin (mm) -58.32051	Ymax (mm) 50	Mesh Was Created In
Zmin (mm)	Zmax (mm) 50	Scaling Factors
View Length Unit In		× 0.001
mm		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.

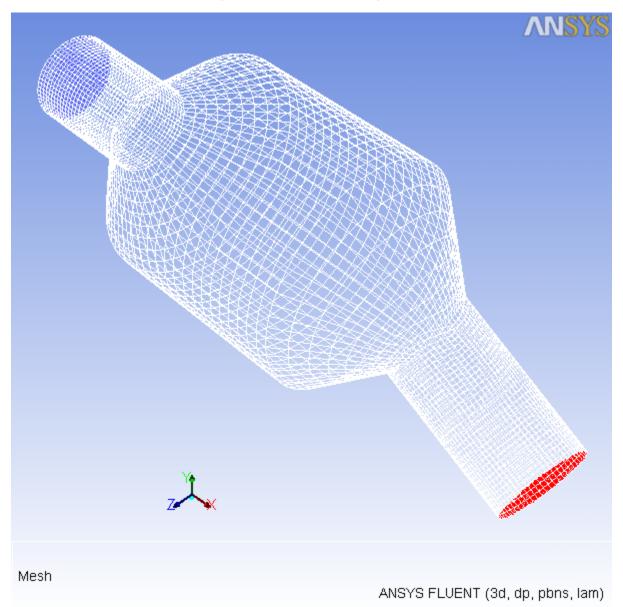
- d. Close the Scale Mesh dialog box.
- 4. Check the mesh.

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 10.2: Mesh for the Catalytic Converter Geometry (p. 449). The hex mesh on the geometry contains a total of 34,580 cells.





10.4.3. General Settings

⇔General

General	
Mesh Scale Check	Report Quality
Display	
Solver	
Type Pressure-Based Density-Based	Velocity Formulation
Time Steady Transient	
Auto-Adjust Solver Defaults	
Gravity	Units
Help	

1. Retain the default solver settings.

10.4.4. Models

1. Select the standard k- ε turbulence model.

 $\mathbf{O}^{\mathsf{Models}} \to \mathbf{\overline{E}}^{\mathsf{Viscous}} \to \mathsf{Edit...}$

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) k-epsilon Model	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1
Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment User-Defined Wall Functions	User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number none TDR Prandtl Number none
Curvature Correction Production Kato-Launder Production Limiter	▼

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.

10.4.5. Materials

1. Add nitrogen to the list of fluid materials by copying it from the **Fluent Database** of materials.



Create/Edit Mat	terials			—
Name		Material Type		Order Materials by
air		fluid		
Chemical Formula		- Fluent Fluid Materials		Chemical Formula
		air	-	Fluent Database
		Mixture		User-Defined Database
		none		r
Properties				
Density (kg/m3)	constant	▼ Edit	Â	
	1.225			
Viscosity (kg/m-s)	[t	▼ Edit		
	constant	▼Edit		
	1.7894e-05		E	
			1	
1			•	
	Change/Create	Delete	e Help	

a. Click the Fluent Database... button to open the Fluent Database Materials dialog box.

Fluent Database Materials		x
Fluent Fluid Materials nitro-methanimynyl (h2cnno2) nitro-methylene (h2cno2) nitro-silane (h3sin) nitrogen (n2) nitrogen-dihydride (nh2) nitrogen-dioxide (no2) Copy Materials from Case Delete	 Material Type fluid Order Materials by Name Chemical Formula 	
Properties Density (kg/m3) Cp (Specific Heat) (j/kg-k)	1.138	4 III
Thermal Conductivity (w/m-k)	constant ▼ View 0.0242	
Viscosity (kg/m-s)	constant ▼ View 1.663e-05	Ŧ
New Edit	Save Copy Close 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.

10.4.6. Cell Zone Conditions

Cell Zone Conditions

Cell Zone Conditions

Zone		
fluid		
substrate		
Phase mixture	Type → fluid →	ID 2
Edit Parameters	Copy Profiles	
Display Mesh		
Porous Formulation Output: Superficial Velo		
Physical Velocit	У	
Help		

- 1. Set the cell zone conditions for the fluid (**fluid**).
 - $\textcircled{Cell Zone Conditions} \rightarrow \overleftarrow{\sqsubseteq} fluid \rightarrow Edit...$

	Fluid								×
	ne Name					_			
1	luid								
Ma	aterial Nam	e nitrogen	▼ Edit						
	_	tion 🔲 Lamina							
	Mesh Moti Porous Zo		Fixed Values						
	_		Iotion Porous Zone Embedde		E	Beaction	Source Terms	Eived Values	Multiphase
		meshive Meshiv	Iouon Porous zone Embedue	eur		Reaction	Source remis	Fixed values [
	Rotation-	Axis Origin		R	ota	tion-Axis Dire	ction		_ ^ _
	X (mm)	0	constant 👻		X	0	constant	•	
	Y (mm)	0	constant 👻		ں آ ۲	0	constant		
	7 ()				l	-			
	Z (mm)	0	constant 🔻		Z [1	constant	•	
									-
	OK 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).

 $\textcircled{Cell Zone Conditions} \rightarrow \overleftarrow{\sqsubseteq} substrate \rightarrow Edit...$

Pluid		×
Zone Name		
substrate		
Material Name nitrogen	▼ Edit	
Frame Motion 🛛 Laminar Zone 🔲 S	Source Terms	
	Fixed Values	
Porous Zone		
Reference Frame Mesh Motion Poro	us Zone Embedded LES Reaction Source	Terms Fixed Values Multiphase
Conical		
Relative Velocity Resistance Formu	lation	•
Viscous Resistance		
Direction-1 (1/m2) 3.846e+07	constant 👻	
Direction-2 (1/m2) 3.846e+10	constant 🗸	
Direction-3 (1/m2) 3.846e+10	constant 👻	
Inertial Resistance		
Alternative Formulation		
Direction-1 (1/m) 20.414	constant 🗸	Ξ
Direction-2 (1/m) 20414	constant 🔹	
Direction-3 (1/m) 20414	constant 💌	
Power Law Model		
		T
	OK Cancel Help	

- 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 10.1: Values for the Principle Direction Vectors (p. 457).

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

Table 10.1: Values for the Principle Direction Vectors

Use the scroll bar to access the fields that are not initially visible in the dialog box.

ii. Enter the values in Table 10.2: Values for the Viscous and Inertial Resistance (p. 457) 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 in the panel.

Table 10.2: 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.

10.4.7. Boundary Conditions

Boundary Conditions

Boundary Conditions

Zone		
default-interior		
default-interior:0	10	
inlet		
outlet		
porous-in		
porous-out substrate-wall		
wall		
l		
Phase	Type	ID
mixture	✓ velocity-inlet	9
	Velocity milet	Ľ
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh	Periodic Conditions	
Highlight Zone		
Help		

1. Set the velocity and turbulence boundary conditions at the inlet (inlet).



Velocity Inlet		
Zone Name		
inlet		
Momentum Thermal Radiation Species	DPM Multiphase U	DS
Velocity Specification Method	Magnitude, Normal to Boun	idary 🔹
Reference Frame	Absolute	▼
Velocity Magnitude (m/s)	22.6	constant 💌
Supersonic/Initial Gauge Pressure (pascal)	0	constant 💌
Turbulence		
Specification Method	ntensity and Hydraulic Diam	eter 🔹
	Turbulent Intensity (%	
	Hydraulic Diameter (mn	n) 42 P
	Cancel Help	

- a. Enter 22.6 m/s for **Velocity Magnitude**.
- b. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.
- 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**).

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{=}$ outlet \rightarrow Edit...

Pressure Outlet
Zone Name
outlet
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method Normal to Boundary
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) 42
OK Cancel Help

- a. Retain the default setting of **0** for **Gauge Pressure**.
- b. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.
- 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).

10.4.8. Solution

1. Set the solution parameters.

Solution Methods

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	
High Order Term Relaxation Options	
Default	
Help	

- 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 residuals during the calculation.

 $\clubsuit Monitors \rightarrow \blacksquare Residuals \rightarrow Edit...$

Residual Monitors					×
Options	Equations				
▼ Print to Console	Residual	Monitor (Check Convergence	Absolute Criteria	_
V Plot	continuity	V		0.001	
Window 1 Curves Axes	x-velocity			0.001	
Iterations to Plot	y-velocity			0.001	
1000	z-velocity			0.001	-
	Residual Values			Convergence Cr	iterion
Iterations to Store	Normalize		Iterations	absolute	•
	Compute Loca	al Scale			
OK Plot Renormalize Cancel Help					

- a. Retain the default settings.
- b. Click **OK** to close the **Residual Monitors** dialog box.
- 3. Enable the plotting of the mass flow rate at the outlet.

Monitors (Surface Monitors) → Create...

💶 Surface Monitor	
Name	Report Type
surf-mon-1	Mass Flow Rate 🔹
Options	Field Variable
V Print to Console	Pressure
V Plot	Static Pressure 👻
Window	Surfaces 🔋 🗏 🗏
2 Curves Axes Write File Name Surf-mon-1.out X Axis Iteration Get Data Every 1 Image Iteration	default-interior default-interior:010 inlet outlet porous-in porous-out substrate-wall wall
Average Over	Highlight Surfaces New Surface ▼
O	K Cancel Help

- a. Enable **Plot** and **Write**.
- b. Select Mass Flow Rate from the Report Type drop-down list.
- c. Select **outlet** in the **Surfaces** selection list.
- d. Click **OK** to close the **Surface Monitor** dialog box.
- 4. Initialize the solution from the inlet.

CSolution Initialization

Solution Initialization	
Initialization Methods O Hybrid Initialization Standard Initialization	
Compute from	
Reference Frame	
Relative to Cell Zone Absolute	
Initial Values	
Gauge Pressure (pascal)	Â
1.455192e-11	
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	
	Ŧ
Initialize Reset Patch	
Reset DPM Sources Reset Statistics	
Help	

a. Select Standard Initialization from the Initialization Methods group box.

Warning

Standard Initialization is the recommended initialization method for porous media simulations. The default **Hybrid initialization** 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 initialization** method can only be used with the **Maintain Constant Velocity Magnitude** option.

- b. Retain the default settings for Standard Initialization method.
- c. Click Initialize once more.
- 5. Save the case file (catalytic_converter.cas.gz).

File \rightarrow Write \rightarrow Case...

6. Run the calculation by requesting 100 iterations.

Run Calculation	
Check Case	Preview Mesh Motion
Pseudo Transient Options	
Fluid Zone	
Time Step Method	Timescale Factor
 User Specified Automatic 	1
Length Scale Method	Verbosity
Conservative -	0
Number of Iterations	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

a. Enter 100 for **Number of Iterations**.

b. Click **Calculate** to begin the iterations.

The solution will converge in approximately 65 iterations. The mass flow rate monitor flattens out, as seen in Figure 10.3: Surface Monitor Plot of Mass Flow Rate with Number of Iterations (p. 465).

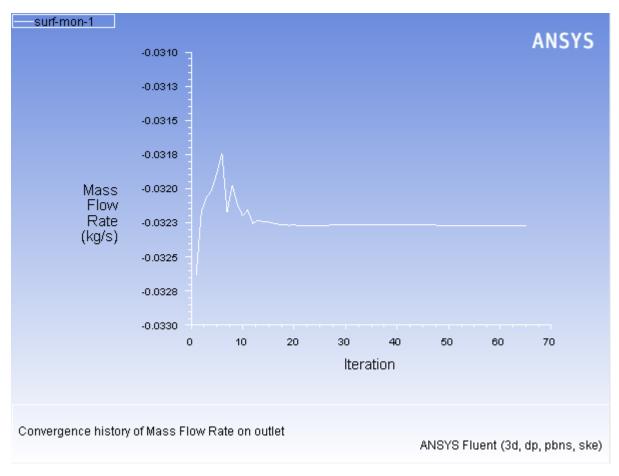


Figure 10.3: Surface Monitor Plot of Mass Flow Rate with Number of Iterations

7. Save the case and data files (catalytic_converter.cas.gz and catalytic_converter.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

Note

If you choose a file name that already exists in the current folder, ANSYS Fluent will prompt you for confirmation to overwrite the file.

10.4.9. Postprocessing

1. Create a surface passing through the centerline for postprocessing purposes.

Surface \rightarrow Iso-Surface...

Iso-Surface		×
Surface of Constant Mesh	 From Surface default-interior default-interior:010 	
Y-Coordinate	▼ inlet	, =
Min (mm) Max (mm) -58.3205 50	outlet porous-in porous-out	-
Iso-Values (mm)	From Zones	
0	fluid	
Image: A marked and the second sec	substrate	
New Surface Name		
y=0		
Create Compute Man	ge 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.

Surface \rightarrow Iso-Surface...

Iso-Surface		×
Surface of Constant Mesh X-Coordinate	From Surface default-interior default-interior:010 inlet	
Min (mm) Max (mm) -1.224606e-15 276.7358 Iso-Values (mm) 95	outlet porous-in porous-out ,	-
New Surface Name x-coordinate-10	fluid substrate	
Create Compute 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.

Surface → Line/Rake...

💶 Line/Rake S	urface	X
Options Une Tool Reset	Type Line 🔻	Number of Points
End Points		
x0 (mm) 95		x1 (mm) 165
y0 (mm) 0		y1 (mm) 0
z0 (mm) 0		z1 (mm) 0
	Select Point	s with Mouse
New Surface Na	me	
porous-d		
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).

• Graphics and Animations $\rightarrow \stackrel{\frown}{=} Mesh \rightarrow Set Up...$

💶 Mesh Displa	у		— X—
Options	Edge Type	Surfaces	
Nodes	🔘 All	porous-in	
Edges	Feature	porous-out	
Faces	Outline	substrate-wall	
Partitions		wall	
Shrink Factor	Feature Angle	x=130	=
0		x=165	
0	20	x=95	-
Surface Name Pa	attern Match	New Surface ▼ Surface Types	
Outline Inter	TIOF	axis dip-surf	Â.
Adjacency		exhaust-fan	
		fan	-
		l <u>.</u> .	÷
Display Colors Close Help			

- a. Disable Edges and enable Faces 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.
- c. Click **Display** and close the **Mesh Display** dialog box.
- 5. Set the lighting for the display.

 \bigcirc Graphics and Animations \rightarrow Options...

Display Options	×
Rendering	Graphics Window
Line Width 1 Point Symbol (+)	Active Window Open 2 Set Color Scheme
Animation Option Wireframe	Workbench -
Double Buffering	Lighting Attributes
Outer Face Culling Hidden Line Removal Hidden Surface Removal Removal Method	 ✓ Lights On Lighting Gouraud ▼
Hardware Z-buffer	Layout
Display Timeout Timeout in seconds 60	 ✓ Titles ✓ Axes ✓ Logo Color White ▼ ✓ Colormap Colormap Alignment Left ▼
Apply Info Lights	Close Help

- a. Disable **Double Buffering** in the **Rendering** group box.
- b. Enable Lights On in the Lighting Attributes group box.
- c. Select **Gouraud** from the **Lighting** drop-down list.
- d. Click Apply and close the Display Options dialog box.
- 6. Set the transparency parameter for the wall zones (substrate-wall and wall).

Graphics and Animations → Scene...

Scene Description		—
Names	Geometry Attributes Type Group Display Transform Iso-Value Pathlines Time Step	Scene Composition Overlays Draw Frame Frame Options
App	Oly 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	X
Geometry Name Group Visibility Visible Visib	Colors Color face-color 255 Red 255 Green 255 Blue 70 Transparency
Apply	Close Help

- i. Ensure that 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 10.4: Velocity Vectors on the y=0 Plane (p. 472)).

Graphics and Animations → $\overline{\equiv}$ Vectors → Set Up...

r			
Vectors			×
Options	Vectors of		
🔽 Global Range	Velocity		•
Auto Range	Color by		
Clip to Range	Velocity		•
Draw Mesh	Velocity Magnitude 🗸		
Style	Min (m/s)	Max (m/s)	
arrow 👻	0.08021907	32.17956	
Scale Skip	Surfaces		
5 1	wall		*
Vector Options	x-coordinate-9		
	x=130 x=165		
Custom Vectors	x=95		E
	y=0		-
Surface Name Pattern			
Match	New Surface 🔻		
	Surface Types		
	axis		
	clip-surf		
	exhaust-fan fan		
			*
Display	Compute Close	Help	

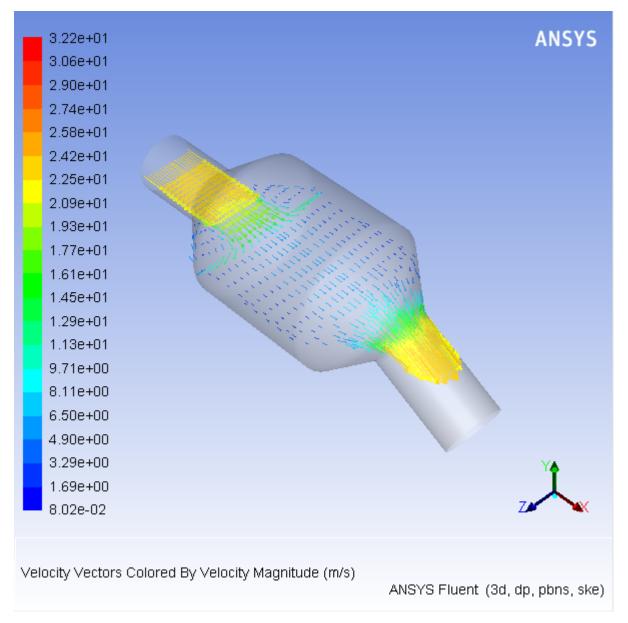
a. Enable **Draw Mesh** in the **Options** group box to open the **Mesh Display** dialog box.

💶 Mesh Displa	iy		—X —
Options	Edge Type	Surfaces	
Nodes		porous-in	
Edges	Feature	porous-out	
Faces	Outline	substrate-wall	
Partitions		' wall	
Shrink Factor	Feature Angle	x=130	E
		x=165	
0	20	x=95	-
Surface Name Pa	attern Match	New Surface Surface Types	
Outline Inter	rior	axis	
		clip-surf	
Adjacency		exhaust-fan	
		fan -	Ψ.
Display Colors Close Help			

- i. Ensure 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.





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 10.5: Contours of Static Pressure on the y=0 plane (p. 474)).

Contours		×
Options	Contours of	
Filled	Pressure	
Vode Values Global Range	Static Pressure	•
Auto Range	Min (pascal)	Max (pascal)
Clip to Range	-398.9136	649.9774
Draw Profiles	Surfaces	
	x-coordinate-9	•
Levels Setup	x=130 x=165	
20 1	x=165 x=95	
	y=0	-
Surface Name Pattern	New Surface	
Match	Surface Types	
	axis	*
	clip-surf	
	exhaust-fan	
	fan	T

Graphics and Animations → \equiv Contours → Set Up...

- a. Enable Filled in the Options group box.
- b. Enable **Draw Mesh** to open the **Mesh Display** dialog box.
 - i. Ensure that **substrate-wall** and **wall** are selected in the **Surfaces** selection list.
 - ii. Click **Display** and close the **Mesh Display** dialog box.
- c. Ensure 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.

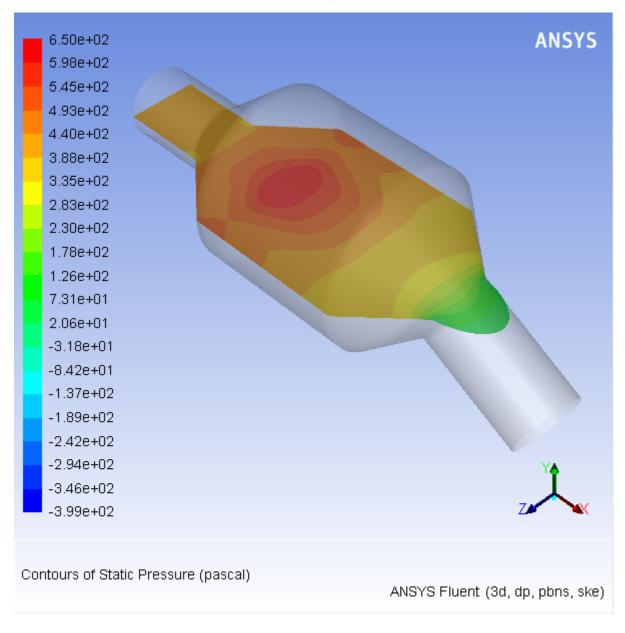


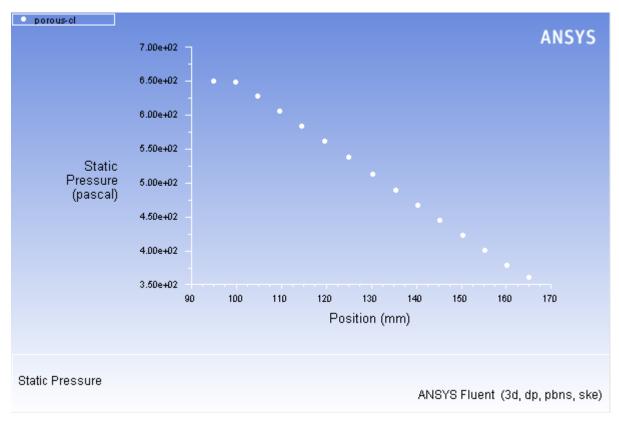
Figure 10.5: Contours of Static Pressure on the y=0 plane

9. Plot the static pressure across the line surface **porous-cl** (Figure 10.6: Plot of Static Pressure on the porouscl Line Surface (p. 475)).

Solution XY Plot			-X
Options Image: Options Image: Option on X Axis Image: Position on Y Axis Image: Order Points	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Pressure Static Pressure X Axis Function Direction Vector Surfaces default-interior default-interior:010 inlet outlet porous-cl porous-in porous-outLatate	
Plot		Curves Close Help	

- a. Ensure 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 10.6: Plot of Static Pressure on the porous-cl Line Surface



As seen in Figure 10.6: Plot of Static Pressure on the porous-cl Line Surface (p. 475), 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 10.7: Contours of the X Velocity on the x=95, x=130, and x=165 Surfaces (p. 477)).

Contours		— ×
Options	Contours of	
V Filled	Velocity	•
Node Values Global Range	X Velocity	•
Auto Range	Min (m/s) Max (m/s)	
Clip to Range	0 7.303799	
Draw Profiles Oraw Mesh	Surfaces	
	x-coordinate-9	
Levels Setup	x=130	
	x=165 x=95	
	y=0	-
Surface Name Pattern	New Surface 🔻	
Match	Surface Types	
	axis	*
	dip-surf exhaust-fan	
	fan	-
	l <u>.</u> .	
Display	Compute Close Help	

• Graphics and Animations $\rightarrow \equiv$ Contours \rightarrow Set Up...

- a. Enable **Filled** in the **Options** group box.
- b. Enable Draw Mesh to open the Mesh Display dialog box.
 - i. Ensure that substrate-wall and wall are selected in the Surfaces selection list.
 - ii. Click **Display** and close the **Mesh Display** dialog box.
- c. Disable Global Range in the Options group box.
- d. Select Velocity... and X Velocity from the Contours of drop-down lists.
- e. Select x=130, x=165, and x=95 in the Surfaces selection list.
- f. Click **Display** and close the **Contours** dialog box.

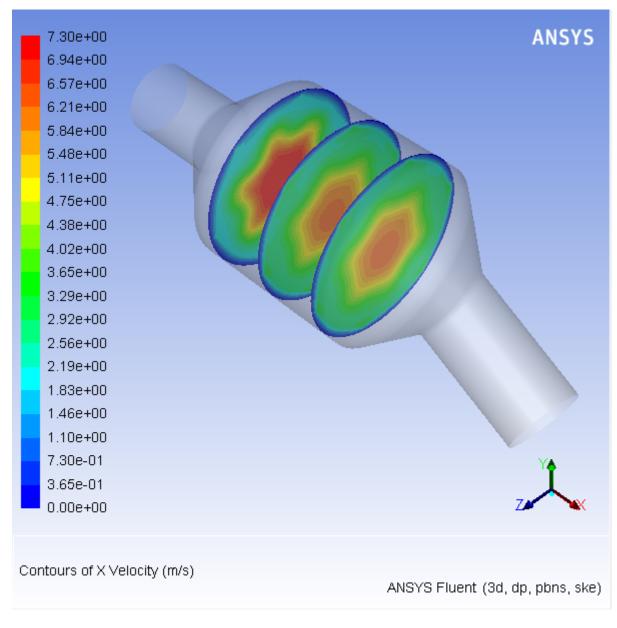


Figure 10.7: Contours of the X Velocity on the x=95, x=130, and x=165 Surfaces

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.

 $\textcircled{Reports} \rightarrow \fbox{Surface Integrals} \rightarrow \texttt{Set Up...}$

Surface Integrals		×
Report Type Mass-Weighted Average	Field Variable	•
Surface Types	X Velocity Surfaces porous-cl porous-in porous-out substrate-wall wall x-coordinate-9 x=130 x=165 x=95 y=0	
Save Output Parameter Compute Write	Highlight Surfaces Mass-Weighted Average (m/s) 4.675797 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=165	4.0394797
	x=95	5.2982397
		A
	Net	4.67579652

Minimum of Facet Values X Velocity	(m/s)
x=165 x=95	2.3443742 0.88216984
Net	0.88216984
Maximum of Facet Values X Velocity	(m/s)
x=165 x=95	6.37113 8.029116
Net	8.0291166

10.5. 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 rapidly discovered through images of velocity vectors and pressure contours. Surface integrals and xy-plots provided purely numeric data.

For additional details about modeling flow through porous media (including heat transfer and reaction modeling), see Porous Media Conditions in the User's Guide.

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. 123).

Chapter 11: Using a Single Rotating Reference Frame

This tutorial is divided into the following sections:

- 11.1. Introduction11.2. Prerequisites11.3. Problem Description11.4. Setup and Solution11.5. Summary
- 11.6. Further Improvements
- 11.7. References

11.1. 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- ε and RNG k- ε 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.

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. 123)

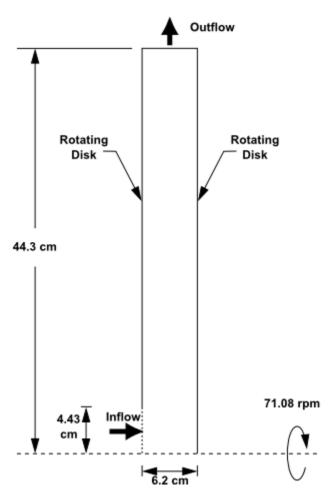
and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

11.3. Problem Description

The problem to be considered is shown schematically in Figure 11.1: Problem Specification (p. 482). 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_{ϕ} . These parameters are defined as follows:

$$C_{w} = \frac{Q}{vr_{out}}$$

$$Re_{\phi} = \frac{\Omega r_{out}^{2}}{v}$$
(11.1)
(11.2)

where Q is the volumetric flow rate, Ω is the rotational speed, v 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 $Re_{\phi} = 10^5$.

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. Solution Using the Standard k- ε Model
- 11.4.9. Postprocessing for the Standard k- ϵ Solution
- 11.4.10. Solution Using the RNG k- ϵ Model
- 11.4.11. Postprocessing for the RNG k- ϵ Solution

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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip single_rotating_R150.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 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 Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Run in Serial under Processing Options.

11.4.2. Mesh

1. Read the mesh file (disk.msh).

 $\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

As ANSYS Fluent reads the mesh file, it will report its progress in the console.

11.4.3. General Settings

1. Check the mesh.

$\textcircled{} General \rightarrow Check$

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.

Note

ANSYS Fluent will issue a warning concerning the high aspect ratios of some cells and possible impacts on calculation of Cell Wall Distance. The warning message includes recommendations for verifying and correcting the Cell Wall Distance calculation. In this particular case the cell aspect ratio does not cause problems so no further action is required. As an optional activity, you can confirm this yourself after the solution is generated by plotting Cell Wall Distance as noted in the warning message.

2. Examine the mesh (Figure 11.2: Mesh Display for the Disk Cavity (p. 485)).

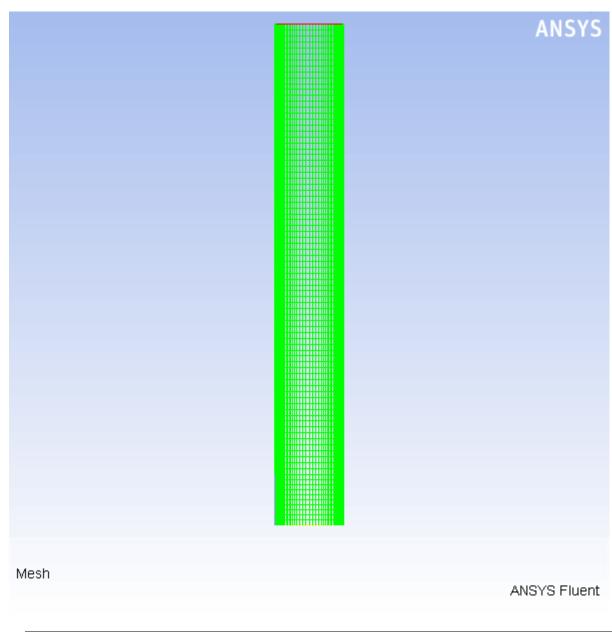


Figure 11.2: Mesh Display for the Disk Cavity

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.

ĢGeneral → 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			×
Quantities		Units	Set All to
force*time-per-volume gas-constant heat-flux heat-flux-resolved heat-generation-rate heat-transfer-coefficient ignition-energy kinematic-viscosity length	•	m cm mm in ft Factor 0.01	default si british cgs
length-inverse length-time-inverse mag-permeability New	+	Offset 0]

- 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.

General

General	
Mesh Scale Display	Check Report Quality
Solver	
Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative
Time ◉ Steady ⓒ Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

- 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 in the 2D Space list.

11.4.4. Models

1. Enable the standard k- ε turbulence model with the enhanced near-wall treatment.

Models →	Edit
----------	-------------

Viscous Model	
Model Inviscid Laminar Spalart-Allmaras (1 eqn) Image: spalart control of the system Image: spalart control of the system	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 User-Defined Functions Turbulent Viscosity None Prandtl Numbers TKE Prandtl Number None TDR Prandtl Number None
Options Production Kato-Launder Production Limiter	-
OK	Cancel Help

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 ϵ -Equation (EWT- ϵ) of the Theory Guide.

11.4.5. Materials

For the present analysis, you will model air as an incompressible fluid with a density of 1.225 kg/m³ 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.

 $\mathbf{O}_{Materials} \rightarrow \mathbf{F}_{air} \rightarrow \mathbf{Create/Edit...}$

ame air		Material Type Ruid	Order Materials by
hemical Formula		Fluent Fluid Materials	Chemical Formula Fluent Database User-Defined Database
roperties Density (kg/m3)	constant 1.225	Edit	
Viscosity (kg/m-s)	constant 1.7894e-05	Edit	

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 User's Guide.

11.4.6. 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.

Cell Zone Conditions

Cell Zone Conditions
Zone
Phase Type ID mixture Tipe 7
Edit Copy Profiles Parameters Operating Conditions Display Mesh Porous Formulation (a) Superficial Velocity
Physical Velocity Help

1. Define the rotating reference frame for the fluid zone (**fluid-7**).

 $\textcircled{Cell Zone Conditions} \rightarrow \overleftarrow{\sqsubseteq} fluid-7 \rightarrow Edit...$

E Fluid	×
Zone Name	
fluid-7	
Material Name air Edit Frame Motion Laminar Zone Source Terms	
Mesh Motion Fixed Values Porous Zone	
Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Multiphase Relative Specification UDF Relative To Cell Zone absolute Zone Motion Function none Rotational Velocity Translational Velocity	
Speed (rpm) 71.08 constant X (m/s) 0 constant Copy To Mesh Motion Y (m/s) 0 constant •	4 M
OK Cancel Help	

- a. Enable Frame Motion.
- b. Click the **Reference Frame** tab.
- c. Enter 71.08 rpm for **Speed** in the **Rotational Velocity** group box.
- d. Click **OK** to close the **Fluid** dialog box.

11.4.7. Boundary Conditions

Boundary Conditions

Boundary Con	ditions	
Zone		
axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6		
Phase mixture	Type ▼ velocity-inlet ▼	ID 2
Edit Parameters Display Mesh	Copy Profiles Operating Conditions Periodic Conditions	
Help		

1. Set the following conditions at the flow inlet (**velocity-inlet-2**).

 $\clubsuit Boundary Conditions \rightarrow \fbox velocity-inlet-2 \rightarrow Edit...$

Velocity Inlet		—
Zone Name		
velocity-inlet-2		
Momentum Thermal Radiation Species	DPM Multiphase U	os
Velocity Specification Method	Components	▼
Reference Frame	Absolute	
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 (rp	0 (mc
Turbulence		
Specification Method	ntensity and Viscosity Ratio	•
	Turbulent Intensity (%	6) 5 P
	Turbulent Viscosity Rati	5 P
ОК	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).

 $\textcircled{} Boundary \ Conditions \rightarrow \fbox{} pressure-outlet-3 \rightarrow Edit...$

Pressure Outlet
Zone Name
pressure-outlet-3
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method From Neighboring Cell
Radial Equilibrium Pressure Distribution
Average Pressure Specification
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Viscosity Ratio
Backflow Turbulent Intensity (%) 5
Backflow Turbulent Viscosity Ratio
OK Cancel 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. Accept the default settings for the disk walls (wall-6).



🞴 Wall							×
Zone Name						-	
wall-6							
Adjacent Cell 2	Zone					_	
fluid-7							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film
Wall Motion		Motion					
 Stationa Moving 	-	√ Relati	ve to Adja	cent Cel	l Zone		
Shear Condit	tion						
No Slip							
Specifie	d Shear rity Coeffi	cient					
Marang		dent					
Wall Roughn	ess						
Roughness	Height (cr	n) 0			constant		-
Roughne	ess Consta	nt 0.5			constant		-
		OK	Canc	el H	lelp		

a. Click **OK** to close the **Wall** dialog box.

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 (i.e., moving with respect to the rotating reference frame). Then you would specify an absolute rotational speed of 0 in the Motion group box.

11.4.8. Solution Using the Standard k- ϵ Model

1. Set the solution parameters.

Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled 🗸	
Spatial Discretization	_
Pressure	*
PRESTO!	
Momentum	Ċ.
Second Order Upwind	
Swirl Velocity	
Second Order Upwind	=
Turbulent Kinetic Energy	
Second Order Upwind 🔹	
Turbulent Dissipation Rate	
Second Order Upwind	÷
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Very Pseudo Transient	
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.

Solution Controls

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	<u>.</u>
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 Setting Pseudo Transient Explicit Relaxation Factors in the User's Guide.

3. Enable the plotting of residuals during the calculation.

 $\clubsuit Monitors \rightarrow \blacksquare Residuals \rightarrow Edit...$

Residual Monitors					-x
Options	Equations				
Print to Console	Residual	Monitor C	heck Convergence	e Absolute Criteria	^
V Plot	continuity		v	0.001	
Window 1 Curves Axes	x-velocity	V	V	0.001	Ē
Iterations to Plot	y-velocity			0.001	-
1000	swirl		\checkmark	0.001	-
	Residual Values			Convergence Cr	riterion
Iterations to Store	Normalize		Iterations	absolute	•
	🔽 Scale				
	Compute Loca	al Scale			
OK Plot	Renormaliz	e Ca	ancel H	elp	

- 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.

 $\clubsuit Monitors (Surface Monitors) \rightarrow Create...$

Surface Monitor	
Name	Report Type
surf-mon-1	Mass Flow Rate 🔹
Options	Field Variable
Print to Console	Pressure v
V Plot	Static Pressure 💌
Window	Surfaces 🔋 🔳 🗏
2 Curves Axes	axis-5
V Write	interior-4
	pressure-outlet-3
File Name	velocity-inlet-2 wall-6
C:/Work/single_rotating/solution_files/suri	
X Axis	
Iteration 🔹	
Get Data Every	
1 Iteration	
Average Over	New Surface 🔻
1	
OK	Cancel Help

a. Enable the **Plot** and **Write** options for surf-mon-1.

Note

When the **Write** option is selected in the **Surface Monitor** dialog box, the mass flow rate history will be written to a file. If you do not enable the **Write** option, the history information will be lost when you exit ANSYS Fluent.

- b. Select Mass Flow Rate from the Report Type drop-down list.
- c. Select pressure-outlet-3 from the Surfaces selection list.
- d. Click **OK** in the **Surface Monitor** dialog box to enable the monitor.
- 5. Initialize the solution.

Solution Initialization

Solution Initialization
Initialization Methods
 Hybrid Initialization Standard Initialization
More Settings Initialize
Reset DPM Sources Reset Statistics
Help

- a. Retain the default selection of Hybrid Initialization from the Initialization Methods group box.
- 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).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}$

7. Start the calculation by requesting 600 iterations.

CRun Calculation

Run Calculation
Check Case Preview Mesh Motion
Pseudo Transient Options
Fluid Zone
Time Step Method Timescale Factor
O User Specified 1 Automatic
Length Scale Method Verbosity Conservative ▼ 0 ▼
Number of Iterations Reporting Interval
Profile Update Interval
Data File Quantities Acoustic Signals
Calculate
Help

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 11.3: Mass Flow Rate History (k- ε Turbulence Model) (p. 501).

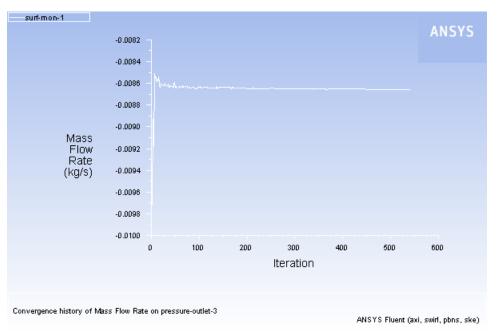


Figure 11.3: Mass Flow Rate History (k- ϵ Turbulence Model)

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 Solving Pseudo-Transient Flow in the Fluent User's Guide.

8. Check the mass flux balance.



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.

Flux Reports		X
Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types axis exhaust-fan fan inlet-vent Boundary Name Pattern Match Save Output Parameter	Boundaries	Results -0.008654586039483547 0.008655219338834286 Image: style="text-align: center;">Image: style="text-align: center;"/>Image: style="text-align:
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 data file (disk-ke.dat.gz).

File \rightarrow Write \rightarrow Data...

Note

If you choose a file name that already exists in the current folder, ANSYS Fluent will prompt you for confirmation to overwrite the file.

11.4.9. Postprocessing for the Standard k- ϵ Solution

1. Display the velocity vectors.



Vectors			×
Options	Vectors of		
🔽 Global Range	Velocity		•
Auto Range	Color by		
Clip to Range	Velocity		•
Draw Mesh	Velocity Magnitude		•
Style	Min	Max	
arrow	0	0	
Scale Skip	Surfaces		
50 1	axis-5		
Vector Options	interior-4 pressure-outlet-3		
	velocity-inlet-2		
Custom Vectors	wall-6		
Surface Name Pattern	New Surface 🔻		
Match			
	Surface Types		
	axis dip-surf		<u>^</u>
	exhaust-fan		
	fan		v
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.

Vector Options	— ×
In Plane Fixed Length X Component Y Component Z Component	Scale Head 0.1 Color
Apply	Close Help

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 11.4: Magnified View of Velocity Vectors within the Disk Cavity (p. 504).

		1	11	1.1					_		×.	3 1 1 1 1 1
-	3.27e+00	111										
	3.10e+00	illi fit.	11	2.5	5	7	1	1		×	×.	
	2.94e+00	111	T_{iN}	χ.,	4	2		1	2	×.	×.	×11∏]
	2.78e+00	111	τ.									
	2.62e+00	Î٦.		<u>`</u>	1	Ĩ.,	Ĩ.,	Ť				
	2.45e+00		1.1	2.5	×	1	\mathcal{F}_{i}	1		N	1	
	2.29e+00		\bar{x}_{∞}	2.2	4	1	1	-	~	\mathbf{x}_{i}	A.	1111
	2.13e+00	111									1	
	1.97e+00	fit.	1	11	1	1	1	1		1	1	
	1.80e+00		1.5	1	\mathbf{x}	\mathbf{z}_{i}	\mathcal{A}_{i}	*	× .	$\sim N_{\odot}$	1	
	1.64e+00	111	52	1.	Γ,	1	1	1	~	N.	1	
	1.48e+00	il.								1	t.	1111
	1.32e+00	ан. -	17	11	1	1	1	1	<u></u>	1	÷.	
	1.15e+00		$\gamma_{\mathcal{A}}$	1.1	$^{\prime}$	\dot{i}	\mathcal{A}_{i}	1	2	1	1	
	9.91e-01	i ltı	14	, ,	Υ.	7	1		1	N	1	
	8.28e-01	1 11,		1	Ť.	÷.					1	t t t l land
	6 660 01		1.1	1.1	1	1	1	1	<u></u>	1	2	
	5.03e-01	i h.	< i	11	ï	1	\mathbf{z}	$\sim 10^{-1}$	18	1	1	
		iliı.	14	Ξ,	ς.	7	÷.,	12	~	N	1	
		f lii.		11	1	1				1	1	1 1 1 449
	1 580 02		1.1	1-1	1	1	1	1	1	÷.	-	t t t l land
		i liı.	14	11	1	1	1	1	~	1	1	

Figure 11.4: Magnified View of Velocity Vectors within the Disk Cavity

Velocity Vectors Colored By Velocity Magnitude (m/s)

ANSYS Fluent (axi, swirl, pbns, ske)

- e. Close the Vectors dialog box.
- 2. Display filled contours of static pressure.



Contours	
Options Filled	Contours of Pressure
 ✓ Node Values ✓ Global Range ✓ Auto Range 	Static Pressure
Clip to Range	0
Draw Mesh	Surfaces
Levels Setup 20 1	interior-4 pressure-outlet-3
Surface Name Pattern Match	New Surface ▼
	Surface Types
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 11.5: Contours of Static Pressure for the Entire Disk Cavity (p. 506). Notice the high pressure that occurs on the right disk near the hub due to the stagnation of the flow entering from the bore.

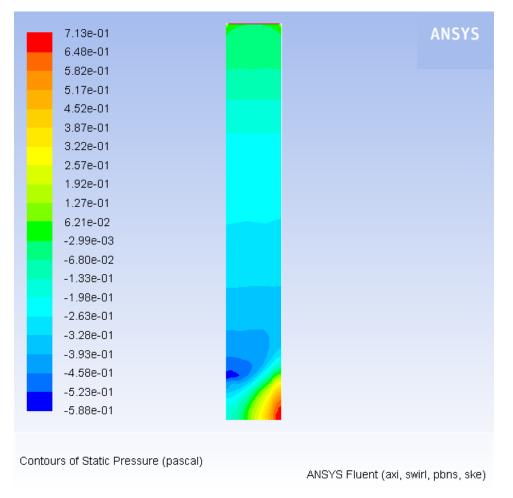


Figure 11.5: Contours of Static Pressure for the Entire Disk Cavity

3. Create a constant *y*-coordinate line for postprocessing.

Surface \rightarrow Iso-Surface...

Iso-Surface	
Surface of Constant Mesh Y-Coordinate Min (cm) Max (cm) 44.3	From Surface
Iso-Values (cm) 37 Image: Second	From Zones E = fluid-7
Create Compute 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 y=37cm for the New Surface Name.
- 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 **y=37cm**.

 $\textcircled{Plots} \rightarrow \overleftarrow{\sqsubseteq} XY Plot \rightarrow Set Up...$

Solution XY Plot		
Options Image: Option of Values Image: Option of Values </td <td>Plot Direction X 1 Y 0 Z 0 Load File Free Data</td> <td>Y Axis Function Velocity Radial Velocity X Axis Function Direction Vector Surfaces axis-5 interior -4 pressure-outlet-3 velocity-inlet-2 wall-6 y=37cm New Surface ▼</td>	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Velocity Radial Velocity X Axis Function Direction Vector Surfaces axis-5 interior -4 pressure-outlet-3 velocity-inlet-2 wall-6 y=37cm 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 **y=37cm** from the **Surfaces** selection list.
- c. Click **Plot**.

Figure 11.6: Radial Velocity Distribution—Standard k- ε Solution(p. 509) shows a plot of the radial velocity distribution along y=37 cm.

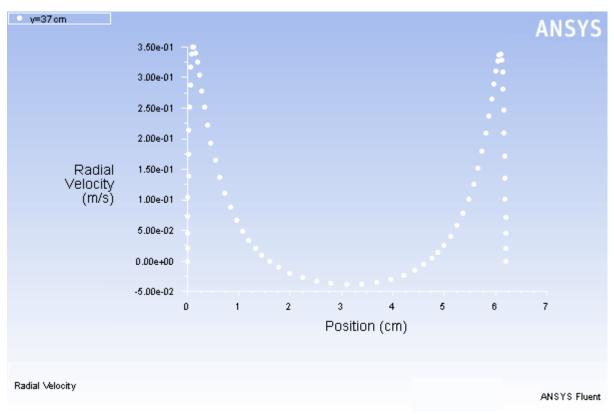


Figure 11.6: Radial Velocity Distribution—Standard k- ϵ Solution

- 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.
- 5. Plot the wall y+ distribution on the rotating disk wall along the radial direction (Figure 11.7: Wall Yplus Distribution on wall-6— Standard k- ε Solution (p. 511)).

$$\mathbf{O}_{\mathsf{Plots}} \to \mathbf{E}_{\mathsf{XY}} \mathsf{Plot} \to \mathsf{Set} \mathsf{Up}_{\mathsf{H}}...$$

Solution XY Plot			×
Options Image: Option of Values Image: Option of Values </td <td>Plot Direction X 0 Y 1 Z 0 Load File Free Data</td> <td>Y Axis Function Turbulence Wall Yplus X Axis Function Direction Vector Surfaces axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6 y=37cm New Surface ▼</td> <td></td>	Plot Direction X 0 Y 1 Z 0 Load File Free Data	Y Axis Function Turbulence Wall Yplus X Axis Function Direction Vector Surfaces axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6 y=37cm 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 **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 a X Y Label	Number Format Type general Precision 3 V	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
	Apply 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 11.7: Wall Yplus Distribution on wall-6— Standard k- ε Solution(p. 511) shows a plot of wall y+ distribution along **wall-6**.

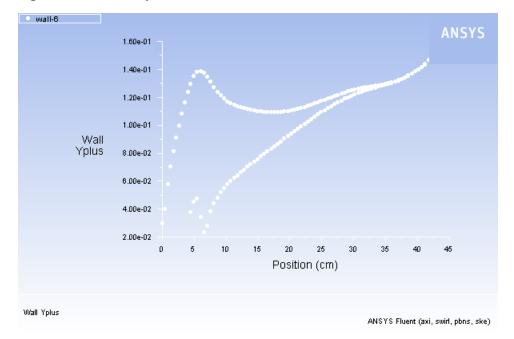


Figure 11.7: Wall Yplus Distribution on wall-6— Standard k- ε Solution

- 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.

11.4.10. Solution Using the RNG k- ϵ Model

Recalculate the solution using the RNG k- ε turbulence model.

1. Enable the RNG k- ε turbulence model with the enhanced near-wall treatment.

♦ Models → Edit...

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 of the 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.

Run Calculation

The solution converges after approximately 220 additional iterations.

3. Save the case and data files (disk-rng.cas.gz and disk-rng.dat.gz).

```
File \rightarrow Write \rightarrow Case & Data...
```

11.4.11. Postprocessing for the RNG k- ϵ Solution

1. Plot the radial velocity distribution for the RNG k- ε solution and compare it with the distribution for the standard k- ε solution.

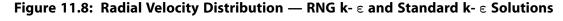
```
 \mathbf{O} \mathsf{Plots} \to \mathbf{F} \mathsf{XY} \mathsf{Plot} \to \mathsf{Set} \mathsf{Up...}
```

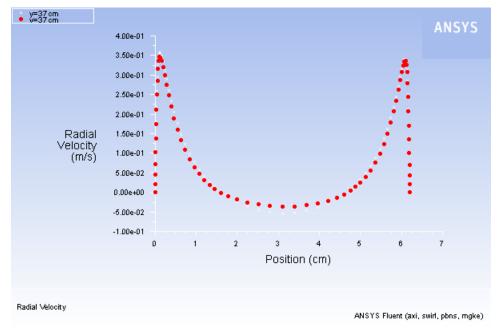
Solution XY Plot			٢
Options Image: Constraint of the system o	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Velocity Radial Velocity X Axis Function Direction Vector Surfaces axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6 y=37cm New Surface ▼	
Plot	Axes	Curves Close Help	

- a. Enter 1 and 0 for **X** and **Y** respectively in the **Plot Direction** group box.
- b. Select Velocity... and Radial Velocity from the Y Axis Function drop-down lists.
- c. Select **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* ε 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- ε data.

Curves -	Solution XY Plot	—		
Curve # 0 • Sample ×	Line Style Pattern Color foreground Weight 1	Marker Style Symbol X Color foreground Size 0.3		
Apply Close Help				

- 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 11.8: Radial Velocity Distribution RNG k- ε and Standard k- ε Solutions (p. 515)).





The peak velocity predicted by the RNG k- ε solution is higher than that predicted by the standard k- ε solution. This is due to the less diffusive character of the RNG k- ε model. Adjust the range of the x axis to magnify the region of the peaks.

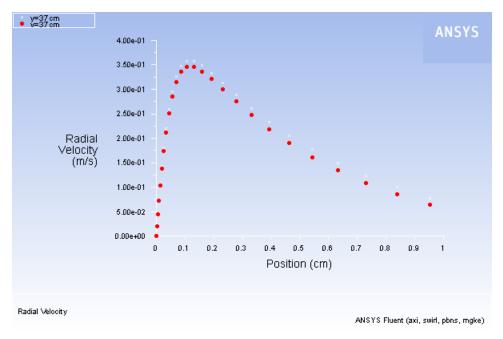
i. Click the **Axes...** button to open the **Axes - Solution XY Plot** dialog box, where you will specify the *x*-axis range.

Axes - Solution XY	Plot	×	
Axis a 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	Minor Rules Color dark gray - Weight 1	
Apply Close Help			

- 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 11.9: RNG k- ε and Standard k- ε Solutions (x=0 cm to x=1 cm)



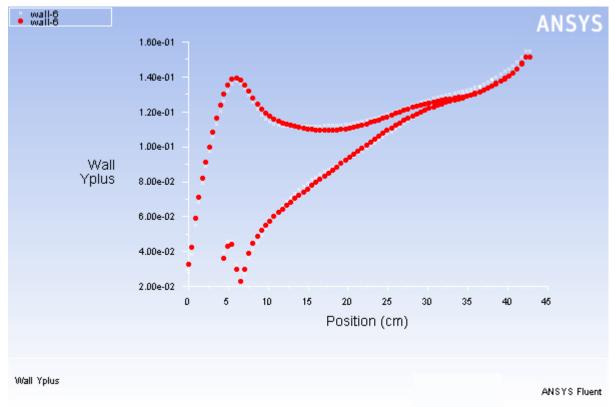
2. Plot the wall y+ distribution on the rotating disk wall along the radial direction Figure 11.10: wall-6 — RNG k- ε and Standard k- ε Solutions (x=0 cm to x=43 cm) (p. 518).

Solution XY Plot		X	
Options Image: Option of Values Image: Option on X Axis Image: Option on Y Axis </td <td>Plot Direction X 0 Y 1 Z 0 Load File Free Data</td> <td>Y Axis Function Turbulence Wall Yplus X Axis Function Direction Vector Surfaces axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6 y=37cm New Surface ▼</td>	Plot Direction X 0 Y 1 Z 0 Load File Free Data	Y Axis Function Turbulence Wall Yplus X Axis Function Direction Vector Surfaces axis-5 interior-4 pressure-outlet-3 velocity-inlet-2 wall-6 y=37cm New Surface ▼	
Plot Axes Curves Close Help			

 $\textcircled{Plots} \rightarrow \overleftarrow{E} XY Plot \rightarrow Set Up...$

- a. Select Turbulence... and Wall Yplus from the Y Axis Function drop-down lists.
- b. Deselect **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* ε 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.





11.5. 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 Model Hierarchy in the User's Guide.

11.6. 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.

```
Mesh \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 non-zero 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. 123).

11.7. 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 12: Using Multiple Reference Frames

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

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, etc.) 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. In other words, 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.

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. 123)

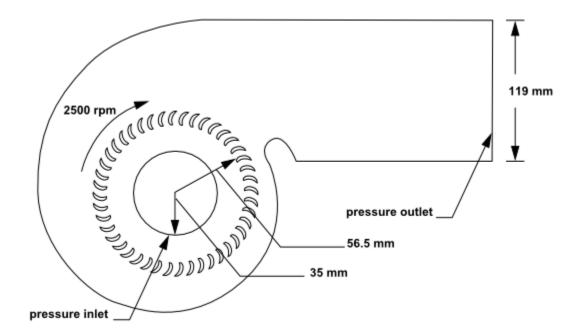
This tutorial also assumes that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

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.

12.3. Problem Description

This problem considers a 2D section of a generic centrifugal blower. A schematic of the problem is shown in Figure 12.1: Schematic of the Problem (p. 522). 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 12.1: Schematic of the Problem



12.4. Setup and Solution

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

- 12.4.1. Preparation
- 12.4.2. Reading and Checking the Mesh and Setting the Units
- 12.4.3. Specifying Solver and Analysis Type
- 12.4.4. Specifying the Models
- 12.4.5. Specifying Materials
- 12.4.6. Specifying Cell Zone Conditions
- 12.4.7. Setting Boundary Conditions
- 12.4.8. Defining Mesh Interfaces
- 12.4.9. Obtaining the Solution
- 12.4.10. Step 9: Postprocessing

12.4.1. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip the multiple_rotating_R150.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** 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 Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the Serial processing option is selected.
- 11. Enable **Double Precision**.

12.4.2. Reading and Checking the Mesh and Setting the Units

1. Read the mesh file (blower-2d.msh).

$\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

The geometry and mesh are displayed in graphics window (Figure 12.2: Mesh of the 2D Centrifugal Blower (p. 525))

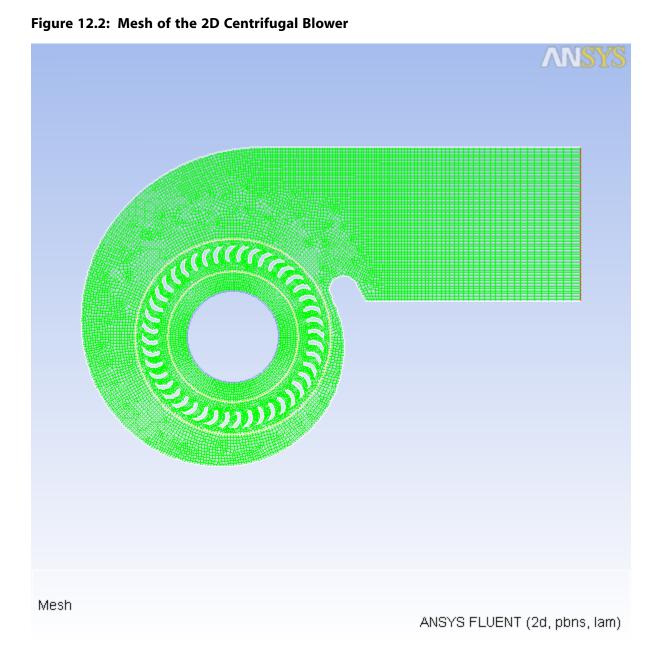
2. Check the mesh.

$\clubsuit General \rightarrow Check$

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.

↓General → Units...

In the problem description, angular velocity is specified in rpm rather than in the default unit of rad/s.

Set Units		—
Quantities acceleration angle angular-velocity area area-inverse collision-rate concentration contact-resistance crank-angle crank-angle crank-angular-velocity density density-gradient	Units rad/s deg/s rpm Factor 0.104719 Offset 0	Set All to default si british cgs 8
New	List Close Help	

- a. Select angular-velocity from the Quantities list and rpm in the Units list.
- b. Close the Set Units dialog box.

12.4.3. Specifying Solver and Analysis Type

1. Retain the default settings of the pressure-based steady-state solver in the Solver group box.

General	
General	
Mesh	
Scale Display	Check Report Quality
Solver	
Type ● Pressure-Based ○ Density-Based	Velocity Formulation
Time ◉ Steady ◎ Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

12.4.4. Specifying the Models

1. Enable the standard k- ε turbulence model.



Viscous Model	
Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) k-epsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Scalable Wall Functions Non-Equilibrium Wall Functions Non-Equilibrium Wall Functions Discrete Wall Treatment User -Defined Wall Functions Pressure Gradient Effects Options Curvature Correction Production Kato-Launder Production Limiter	Model Constants Cmu 0.09 C1-Epsilon = 1.44 = C2-Epsilon = I.92 TKE Prandtl Number 1 - User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number IDR Prandtl Number Inone TDR Prandtl Number Inone
OK	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.

12.4.5. Specifying Materials

1. Retain the default properties for air.



💶 Create/Edit Ma	terials		×
Name air		Material Type fluid	Order Materials by
Chemical Formula		FLUENT Fluid Materials air Mixture none	Chemical Formula FLUENT Database User-Defined Database
Properties			
Density (kg/m3) Viscosity (kg/m-s)	constant 1.225	▼ Edit	
	0.7894e-05	Edit	
		~	
	Change/Create	Delete Close Help	

Extra

If needed, you could modify the fluid properties for air or copy another material from the database.

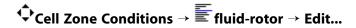
2. Click Close to close the Create/Edit Materials dialog box.

For details, see Physical Properties in the User's Guide.

12.4.6. Speci	fying C	Cell Zone	Conditions
---------------	---------	-----------	------------

Cell Zone Conditions
Zone
fluid-casing fluid-inlet fluid-rotor
Phase Type ID
mixture v fluid v 4
Edit Copy Profiles
Parameters Operating Conditions
Display Mesh
Superficial Velocity
O Physical Velocity
Help

1. Define the boundary conditions for the rotational reference frame (**fluid-rotor**).



Tuid	×
Zone Name	
fluid-rotor	
Material Name air Edit Frame Motion Laminar Zone Source Terms	
Mesh Motion Fixed Values Porous Zone	
Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Multiphase	1
Relative Specification UDF	
Relative To Cell Zone absolute	
Rotation-Axis Origin	
X (m) 0 constant 👻	
Y (m) 0 constant •	Ξ
Rotational Velocity Translational Velocity	
Speed (rpm) -2500 constant X (m/s) 0 constant	
Copy To Mesh Motion Y (m/s) 0 constant	Ţ
OK Cancel Help	

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.

Cell Zone Conditions \rightarrow Display Mesh...

- i. Click the interior zones, **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 the Enter key 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.

12.4.7. Setting Boundary Conditions

Boundary Con	ditions	
Zone		
blades casing default-interior default-interior:0: default-interior:0: inlet interface-1 interface-2 interface-3 interface-4 outlet		
Phase mixture	Type wall	ID 7
Edit Parameters	Copy Profiles Operating Conditions]
Display Mesh	Periodic Conditions	
Help		

1. Set the boundary conditions for the flow inlet (**inlet**) as specified in the problem description (see Figure 12.1: Schematic of the Problem (p. 522)).



Pressure Inlet	×
Zone Name	
inlet	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Reference Frame Absolute	•
Gauge Total Pressure (pascal)	•
Supersonic/Initial Gauge Pressure (pascal) 0 constant	•
Direction Specification Method Normal to Boundary	-
Turbulence	
Specification Method Intensity and Viscosity Ratio	-
Turbulent Intensity (%) 5	e
Turbulent Viscosity Ratio 10	P
OK 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 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.

💶 Wall				×
Zone Name				
blades				
Adjacent Cell Zone				
fluid-rotor				
Momentum Thermal	Radiation Species	DPM Multipha	se UDS Wall Film	
Wall Motion	Motion			
 Stationary Wall Moving Wall 	 Relative to Adja Absolute 	cent Cell Zone	Speed (rpm)	P
			Rotation-Axis Origin	
	 Translational Rotational 		X (m) 0	
	Components		Y (m)	e
			0	P
Shear Condition				
 No Slip Specified Shear Specularity Coeffici Marangoni Stress 	ient			
Wall Roughness				
Roughness Height (m)	0	constant		
Roughness Constant	0.5	constant	~	
	ОК	Cancel	Help	

With *fluid-rotor* set to a rotating reference frame, *blades* becomes a moving wall.

a. Select Moving Wall in the Wall Motion list.

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.

12.4.8. 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**.

Create/Edit Mesh Interfaces		×
Mesh Interface	Interface Zone 1	Interface Zone 2
int1	interface-1	interface-2
Ξ		
	interface-1	interface-1
	interface-2	interface-2
	interface-3	interface-3
	interface-4	interface-4
Interface Options	Boundary Zone 1	Interface Wall Zone 1
Periodic Boundary Condition		
Periodic Repeats	1 2 - 1 - 2 - 2	
Coupled Wall	Boundary Zone 2	Interface Wall Zone 2
Matching		
		Interface Interior Zone
Periodic Boundary Condition		1
Type Offset		
Translational X (m) Rotational	Y (m) 0	
✓ Auto Compute Offset		

♦ Mesh Interfaces → Create/Edit...

- 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.

12.4.9. Obtaining the Solution

1. Set the solution parameters.

♀Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled 🗸	
Spatial Discretization	
Gradient	Â.
Least Squares Cell Based 🔹	
Pressure	
Second Order 🔹	
Momentum	=
Second Order Upwind	
Turbulent Kinetic Energy	
Second Order Upwind	
Turbulent Dissipation Rate	
Second Order Upwind	Ŧ
Transient Formulation	
	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Default	
Help	

- a. Select **Coupled** from **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.

 $\diamondsuit \mathsf{Monitors} \to \mathbf{\overline{\overline{\mathbb{F}}}} \mathsf{Residuals} \to \mathsf{Edit}...$

Residual Monitors					×
Options	Equations				_
Print to Console	Residual	Monitor C	Check Convergence	Absolute Criteria	*
Vert Plot	continuity			5e-5	
Window 1 Curves Axes	x-velocity			0.001	E
Iterations to Plot	y-velocity			0.001	
1000	k			0.001	-
	Residual Values			Convergence Cr	riterion
Iterations to Store	Normalize		Iterations	absolute	•
	Scale				
	Compute Loca	al 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 monitor and plot the volume flow rate at the flow outlet.

Monitors (Surface Monitors) → Create...

Surface Monitor	
Name	Report Type
surf-mon-1	Volume Flow Rate 🔹
Options	Field Variable
Print to Console	Pressure
V Plot	Static Pressure 💌
Window	Surfaces
2 Curves Axes	inlet 🔹
V Write	interface-1
	interface-2
File Name	interface-3
surf-mon-1.out	interface-4
	interior-16
X Axis	interior-19
Iteration 👻	outlet
Cat Data Every	wall-17
Get Data Every	wall-18
1 Iteration V	wall-20
Average Over(Iterations)	wall-21
	New Surface 💌
1	
0	Cancel Help

a. Enable the **Plot** and **Write** options for surf-mon-1.

Note

When the **Write** option is selected in the **Surface Monitor** dialog box, the mass flow rate history will be written to a file. If you do not enable the **Write** option, the history information will be lost when you exit ANSYS Fluent.

- b. Select Volume Flow Rate from the Report Type drop-down list.
- c. Select **outlet** from the **Surfaces** selection list.
- d. Click **OK** in the **Surface Monitor** dialog box to enable the monitor.

4. Initialize the solution.

Solution Initialization

Solution Initialization			
Initialization Methods			
 Hybrid Initialization Standard Initialization 			
More Settings Initialize			
Patch			
Reset DPM Sources Reset Statistics			
Help			

- a. Retain the default selection of **Hybrid Initialization** in the **Initialization Methods** group box.
- 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).

File \rightarrow Write \rightarrow Case...

6. Start the calculation by requesting 150 iterations.

CRun Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Number of Iterations	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

a. Enter 150 for **Number 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 95 iterations (when all residuals have dropped below their respective criteria).

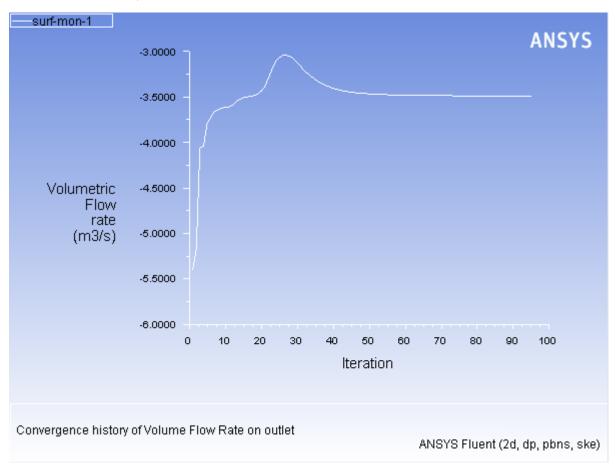


Figure 12.3: History of Volume Flow Rate onoutlet

The surface monitor 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 m^3 /s.

Note

You can examine the residuals history by selecting it from the graphics window dropdown list.

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.

12.4.10. Step 9: Postprocessing

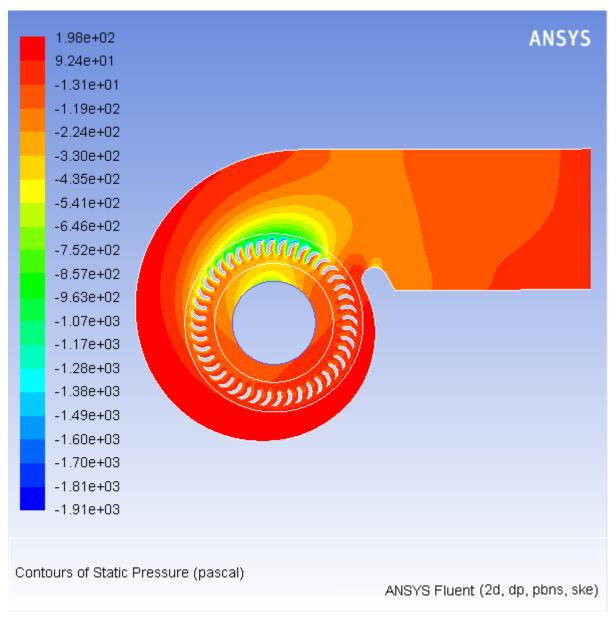
1. Display filled contours of static pressure (Figure 12.4: Contours of Static Pressure (p. 542)).

=

Graphics and Anii	mations $\rightarrow \equiv$ Contours \rightarrow Set Up
Contours	
Options	Contours of
Filled	Pressure
 Global Range Auto Range Clip to Range 	Min (pascal) Max (pascal) -1912.077 197.8676
Draw Profiles	Surfaces
Levels Setup 20 1	blades casing default-interior:013 default-interior:015
Surface Name Pattern	New Surface Surface Types
	axis dip-surf exhaust-fan fan
Display	Compute Close Help

- a. Enable **Filled** in the **Options** group box.
- b. Select Pressure... and Static Pressure from the Contours of drop-down lists.
- c. Click **Display** and close the **Contours** dialog box (see Figure 12.4: Contours of Static Pressure (p. 542)).
 Pressure distribution in the flow domain is plotted in graphics window.





2. Display absolute velocity vectors (Figure 12.5: Velocity Vectors (p. 544)).

Graphics and Animations → $\boxed{=}$ Vectors → Set Up...

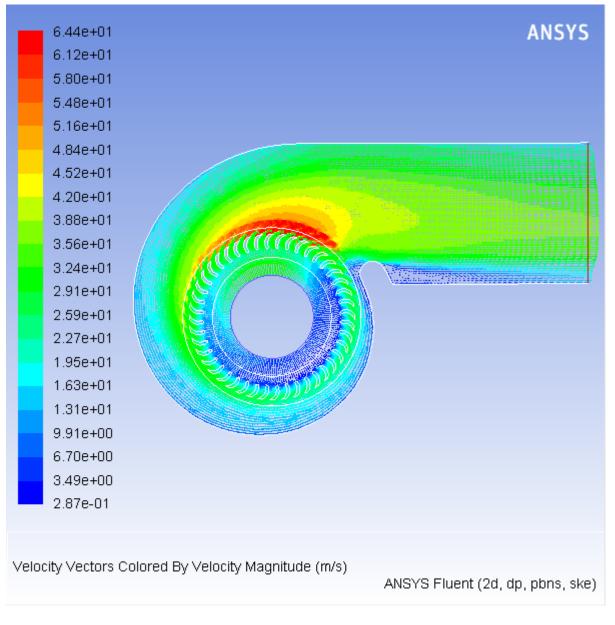
Vectors			×
Options	Vectors of		
🔽 Global Range	Velocity		•
Auto Range	Color by		
Clip to Range	Velocity		•
Draw Mesh	Velocity Magnitude 🗸		
Style	Min (m/s)	Max (m/s)	
arrow 🗸	0.2869741	64.42094	
Scale Skip	Surfaces		
10 0	blades		*
Vector Options	casing default-interior		E
	default-interior:013		
Custom Vectors	default-interior:015		
	inlet		-
Surface Name Pattern			
Match	New Surface 🔻		
	Surface Types		
	axis		<u> </u>
	clip-surf exhaust-fan		
	fan		-
	[I= .		
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 12.5: Velocity Vectors (p. 544)).

Figure 12.5: 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).

Graphics and Animations → \blacksquare Vectors → Set Up...

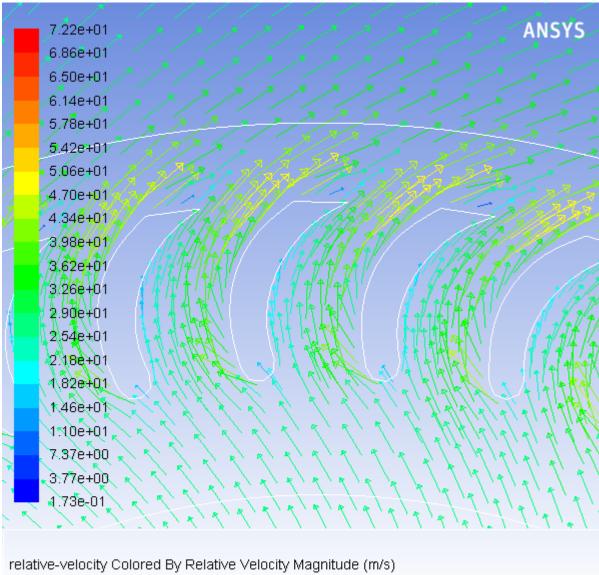
-			
Vectors			×
Options	Vectors of		
Global Range	Relative Velocity		-
Auto Range	Color by		
Clip to Range	Velocity		
Auto Scale			
Draw Mesh	Relative Velocity Mag		•
Style	Min (m/s)	Max (m/s)]
arrow 🔻	0.0580698	60.29894	
Scale Skip	Surfaces		
2 0	blades		
	casing		-
Vector Options	default-interior		=
Custom Vectors	default-interior:013		
Custom vectors	default-interior:015		
	inlet		-
Surface Name Pattern	New Surface -		
Match	INEW SUFFACE		
	Surface Types		
	axis		
	clip-surf		
	exhaust-fan fan		
			Ψ.
Display	Compute Close	Help	

- a. In the Reference Values task page, select fluid-rotor from the Reference Zone drop-down list.
- b. In the Vectors dialog box, select Relative Velocity from the Vectors of drop-down list.
- c. Select Velocity... and Relative Velocity Magnitude from the Color by drop-down list.
- d. Set Scale to 2.
- e. 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.

f. Zoom in on the rotor blade region as shown in Figure 12.6: Relative Velocity Vectors (p. 546) and examine the air flow through the rotor blade passages.





ANSYS Fluent (2d, dp, pbns, ske)

4. Report the mass flux at **inlet** and **outlet**.

 $\textcircled{P} Reports \rightarrow \overleftarrow{E} Fluxes \rightarrow Set Up...$

r		
E Flux Reports		—
Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types	Boundaries Casing default-interior default-interior:013 default-interior:015 inlet	Results
axis exhaust-fan fan inlet-vent Boundary Name Pattern Match	interface-1	-4.273831059660111
Save Output Parameter	Close He	Net Results (kg/s) -8.861845e-06

- 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.

 $\clubsuit Reports \rightarrow \overleftarrow{E} Surface Integrals \rightarrow Set Up...$

12.5. 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 ANSYS Fluent User's Guide for details.

12.6. 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. 123).

Chapter 13: Using the Mixing Plane Model

This tutorial is divided into the following sections:

- 13.1. Introduction 13.2. Prerequisites
- 13.3. Problem Description
- 13.4. Setup and Solution
- 13.5. Summary
- 13.6. Further Improvements

13.1. Introduction

This tutorial considers the flow in an axial fan with a rotor in front and stators (vanes) in the rear. This configuration is typical of a single-stage axial flow turbomachine. By considering the rotor and stator together in a single calculation, you can determine the interaction between these components.

This tutorial demonstrates how to do the following:

- Use the standard k- ε model with enhanced wall treatment.
- Use a mixing plane to model the rotor-stator interface.
- Calculate a solution using the pressure-based solver.
- Compute and display circumferential averages of total pressure on a surface.

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. 123)

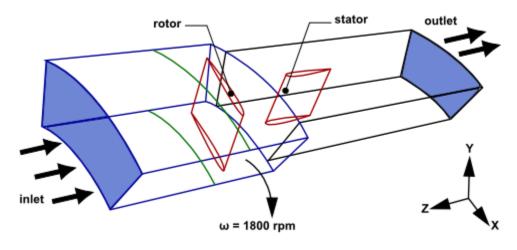
and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

13.3. Problem Description

The problem to be considered is shown schematically in Figure 13.1: Problem Specification (p. 550). The rotor and stator consist of 9 and 12 blades, respectively. A steady-state solution for this configuration using only one rotor blade and one stator blade is desired. Since the periodic angles for the rotor and stator are different, a mixing plane must be used at the interface.

The mixing plane is defined at the rotor outlet / stator inlet. The mesh is set up with periodic boundaries on either side of the rotor and stator blades. A pressure inlet is used at the upstream boundary and a pressure outlet at the downstream boundary. Ambient air is drawn into the fan (at 0 Pa gauge total pressure) and is exhausted back out to the ambient environment (0 Pa static pressure). The hub and blade of the rotor are assumed to be rotating at 1800 rpm.

Figure 13.1: Problem Specification



13.4. Setup and Solution

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

13.4.1. Preparation
13.4.2. Mesh
13.4.3. General Settings
13.4.4. Models
13.4.5. Mixing Plane
13.4.6. Materials
13.4.7. Cell Zone Conditions
13.4.8. Boundary Conditions
13.4.9. Solution
13.4.10. Postprocessing

13.4.1. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip mixing_plane_R150.zip to your working folder.

The file fanstage.msh can be found in the mixing_plane directory created after unzipping the file.

Copy the fanstage.msh file into your working directory.

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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Run in single precision (disable Double Precision).
- 11. Ensure Serial is selected under Processing Options.

13.4.2. Mesh

1. Read the mesh file fanstage.msh.

```
\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}
```

As ANSYS Fluent reads the mesh file, it will report its progress in the console.

13.4.3. General Settings

1. Check the mesh.

$\mathbf{O}_{General} \rightarrow \mathbf{Check}$

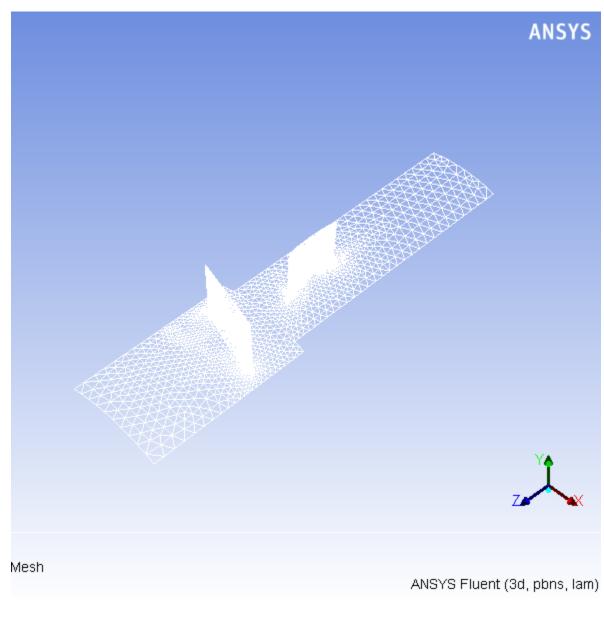
ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number. You will notice that ANSYS Fluent issues several warning messages concerning translation vectors with suggestions to check periodic setup. These arise because you have not yet specified the periodicity for zones 11 and 22. You can ignore these warnings because you will specify the periodicity in a later step.

2. Display the mesh (Figure 13.2: Mesh Display for the Multistage Fan (p. 553)).

 $\clubsuit General \rightarrow Display...$

💶 Mesh Displa	у		×
Options	Edge Type	Surfaces	
Nodes	All	rotor-blade	
Edges	Feature	rotor-hub	
Faces	Outline	rotor-inlet-hub	
Partitions		rotor-inlet-shroud	_
Chuich Easter	r Faaluwa Aaala	rotor-shroud	E
	Feature Angle	stator-blade	
0	20	stator-hub	T
Surface Name Pa	Match	New Surface Surface Types axis clip-surf exhaust-fan fan	
Display Colors Close Help			

- a. Select only **rotor-blade**, **rotor-hub**, **rotor-inlet-hub**, **stator-blade**, and **stator-hub** from the **Surfaces** selection list.
- b. Click **Display** and close the **Mesh Display** dialog box.





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.

3. Retain the default solver settings.

⇔General

General
Mesh Scale Check Report Quality Display
Solver
Type Velocity Formulation Image: Pressure-Based Image: Pressure-Based Image: Density-Based Image: Pressure-Based
Time ◎ Steady ○ Transient
Gravity Units
Help

4. Define new units for angular velocity.

General → Units...

The angular velocity for this problem is known in rpm, which is not the default unit for angular velocity. You will need to redefine the angular velocity units as rpm.

Set Units			×
Quantities acceleration angle angular-velocity area area-inverse collision-rate concentration contact-resistance crank-angle crank-angle crank-angular-velocity density density-gradient	-	Units rad/s deg/s rpm Factor 0.1047198 Offset 0	Set All to default si british cgs
	New List	Close Help	

- a. Select angular-velocity from the Quantities selection list and rpm from the Units selection list.
- b. Close the Set Units dialog box.

13.4.4. Models

1. Enable the standard k- ε turbulence model with enhanced wall treatment.

�Models → E Viscou	ıs → Edit
---------------------	-----------

Viscous Model	—
Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) k-omega (2 eqn) Transition k-kl-omega (3 eqn)	Model Constants Cmu 0.09 C1-Epsilon 1.44
 Transition SST (4 eqn) Reynolds Stress (7 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) Large Eddy Simulation (LES) 	C2-Epsilon 1.92 TKE Prandtl Number 1
k-epsilon Model Standard RNG Realizable Near-Wall Treatment	User-Defined Functions Turbulent Viscosity none Prandtl Numbers
 Standard Wall Functions Scalable Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment User-Defined Wall Functions Enhanced Wall Treatment Options Enhanced Wall Treatment Options	TKE Prandtl Number none TDR Prandtl Number none
Options Curvature Correction Production Kato-Launder Production Limiter	
OK	Cancel Help

a. Select **k-epsilon (2 eqn)** from the **Model** list.

The Viscous Model dialog box will expand.

- b. Retain the default selection of **Standard** from the **k-epsilon Model** list.
- c. Select Enhanced Wall Treatment from the Near-Wall Treatment list.
- d. Click **OK** to close the **Viscous Model** dialog box.

13.4.5. Mixing Plane

Define → **Mixing Planes...**

In this step, you will create the mixing plane between the pressure outlet of the rotor and the pressure inlet of the stator.

Mixing Planes		
Mixing Plane	Upstream Zone pressure-outlet-rotor	Downstream Zone pressure-inlet-stator
	pressure-outlet-rotor pressure-outlet-stator	pressure-inlet-rotor pressure-inlet-stator
Mixing Plane Geometry Radial Axial	Interpolation Points 10	Global Parameters Averaging Method Under-Relaxation Area
		Mass Default Apply
	Create Delete Close	Help

- 1. Select pressure-outlet-rotor from the Upstream Zone selection list.
- 2. Select pressure-inlet-stator from the Downstream Zone selection list.
- 3. Retain the selection of Area from the Averaging Method list.
- 4. Click Create and close the Mixing Planes dialog box.

ANSYS Fluent will name the mixing plane by combining the names of the zones selected as the **Upstream Zone** and **Downstream Zone**. This new name will be displayed in the **Mixing Plane** list.

The essential idea behind the mixing plane concept is that each fluid zone (stator and rotor) is solved as a steady-state problem. At some prescribed iteration interval, the flow data at the mixing plane interface is averaged in the circumferential direction on both the rotor outlet and the stator inlet boundaries. ANSYS Fluent uses these circumferential averages to define "profiles" of flow properties. These profiles are then used to update boundary conditions along the two zones of the mixing plane interface.

In this example, profiles of averaged total pressure (p_{0}) , static pressure (p_{c}) , direction cosines of the local

flow angles in the radial, tangential, and axial directions $(\alpha_r, \alpha_t, \alpha_z)$, turbulent kinetic energy (k), turbulent dissipation rate (ε) , and radius (r) are computed at the rotor exit and used to update boundary conditions at the stator inlet. Likewise, the same profiles (except for that of total pressure) are computed at the stator inlet and used as a boundary condition on the rotor exit.

The default method for calculating mixing plane profiles uses an area-weighted averaging approach. This method allows reasonable profiles of all variables to be created, regardless of the mesh topology. In some cases, a mass flow-weighted averaging may be appropriate (for example, with compressible turbomachinery flows). For such cases, **Mass** should be selected from the **Averaging Method** list. A third averaging approach (the **Mixed-Out** average) is also available for flows with ideal gases. Refer to Choosing an Averaging Method in the Theory Guide for more information on these averaging methods.

You can view the profiles computed at the rotor exit and stator inlet in the **Profiles** dialog box.

$\textbf{Define} \rightarrow \textbf{Profiles...}$

Profiles		
Profile pressure-outlet-rotor pressure-inlet-stator	Fields p0 p ni nj nk ni2 nj2 nk2 k e r	Interpolation Method Constant Inverse Distance Least Squares
Delete Orient Rea	ad Write Apply C	lose Help

You will also see that these profiles appear in the boundary conditions dialog boxes for the rotor exit and stator inlet.

For more information on mixing planes, see The Mixing Plane Model in the User's Guide.

13.4.6. Materials

1. Retain the default properties for air.

lame		Material Type	Order Materials by
air		fluid	Name
hemical Formula		Fluent Fluid Materials	Chemical Formula
		air	Fluent Database
		Mixture	User-Defined Database.
		none	-
roperties			
Density (kg/m3)	constant	Edit	
	1.225		
Viscosity (kg/m-s)	constant	Edit	
	1.7894e-05		
		E	
		*	

a. Review the default properties for air.

For the present analysis, you will model air as an incompressible fluid with a density of 1.225 kg/m³ and a dynamic viscosity of 1.7894×10^{-5} kg/m-s. Since these are the default values, no change is required in the **Create/Edit Materials** dialog box.

b. Close the Create/Edit Materials dialog box.

13.4.7. Cell Zone Conditions

Cell Zone Conditions

Cell Zone Conditions

Zone			
fluid-rotor			
fluid-stator			
Phase	Type		ID
mixture	- fluid	•	13
Edit	Copy	Profiles	
Parameters	Operating (Conditions	
Display Mesh			
Porous Formulation Superficial Velo			
O Physical Velocit			
- Hyaical Velocit	Ŷ		
Help			

1. Set the conditions for the rotor fluid (**fluid-rotor**).

 $\textcircled{Cell Zone Conditions} \rightarrow \fbox{fluid-rotor} \rightarrow \texttt{Edit...}$

Fluid							
ne Name			-				
luid-rotor							
Mesh Motion Porous Zone	Edit Laminar Zone Source Terms Fixed Values						
Reference Frame	Mesh Motion Porous Zone Emb	edded LES	Reaction	Source Te	erms Fixed Value	es Multiphase	
Relative Specifica	ition	UDF					-
Relative To Cell	Zone absolute	Zone M	otion Fund	tion none		•	
Rotation-Axis Ori	gin	Rot	ation-Axis	Direction			
X (m) 0	constant 👻) x	0	constant 👻		•	
Y (m)	constant 🗸) Y	0	cons	stant	•	
Z (m) 0	constant 🗸) z	-1	cons	stant	•	:
Rotational Velocit	у		ranslation	al Velocity			
Speed (rpm)	800 constant	-	X (m/s)	0	constant	•	
Copy To Mesh M	lotion		Y (m/s)	0	constant	•	
			Z (m/s)	0	constant	-	
		L					
	OK	Cance	el Hel	p			

- a. Enable Frame Motion.
- b. Enter -1 for **Z** in the **Rotation-Axis Direction** group box.

According to the right-hand rule (see Figure 13.1: Problem Specification (p. 550)), the axis of rotation is the -Z axis.

- c. Enter 1800 rpm for **Speed** in the **Rotational Velocity** group box.
- d. Click **OK** to close the **Fluid** dialog box.
- 2. Set the conditions for the stator fluid (**fluid-stator**).

 $\textcircled{Cell Zone Conditions} \rightarrow \overleftarrow{\sqsubseteq} fluid-stator \rightarrow Edit...$

E Fluid
Zone Name
fluid-stator
Material Name air
Frame Motion Laminar Zone Source Terms Fixed Values
Porous Zone
Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Multiphase
Rotation-Axis Origin Rotation-Axis Direction
X (m) 0 constant V 0 constant V
Y (m) 0 constant V 0 constant V
Z (m) 0 constant V Z -1 constant V
-
OK Cancel Help

- a. Enter -1 for **Z** in the **Rotation-Axis Direction** group box. Even though **Frame Motion** is not enabled for the fluid-stator zone, the rotational axis specification must be consistent between zones.
- b. Click **OK** to close the **Fluid** dialog box.

13.4.8. Boundary Conditions

Boundary Conditions

Boundary Conditions	
Zone	
interior-16 interior-3	-
periodic-11 periodic-22 pressure-inlet-rotor pressure-outlet-rotor pressure-outlet-rotor pressure-outlet-stator rotor-blade rotor-hub rotor-inlet-hub rotor-inlet-shroud rotor-shroud stator-blade stator-blade stator-hub stator-shroud	4 III
Phase Type ID mixture v periodic v 11	
Edit Copy Profiles Parameters Operating Conditions Display Mesh Periodic Conditions Highlight Zone	
Help	

1. Specify rotational periodicity for the periodic boundary of the rotor (**periodic-11**).

Periodic	X
Zone Name	
periodic-11	
	Periodic Type Translational Rotational
	OK Cancel Help

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{=}$ periodic-11 \rightarrow Edit...

- a. Select Rotational from the Periodic Type list.
- b. Click **OK** to close the **Periodic** dialog box.
- 2. Specify rotational periodicity for the periodic boundary of the stator (periodic-22).

 $\textcircled{P} Boundary \ Conditions \rightarrow \fbox{Periodic-22} \rightarrow \texttt{Edit...}$

Periodic	×
Zone Name periodic-22	
	Periodic Type Translational Rotational
	OK Cancel Help

- a. Select Rotational from the Periodic Type list.
- b. Click **OK** to close the **Periodic** dialog box.
- 3. Set the conditions for the pressure inlet of the rotor (pressure-inlet-rotor).

♀ Boundary Conditions → $ ilde{ extsf{eq}}$ pre	ssure-inlet-rotor \rightarrow E	Edit
Pressure Inlet		•••
Zone Name		
pressure-inlet-rotor		
Momentum Thermal Radiation Species	B DPM Multiphase U	os
Reference Frame	Absolute	•
Gauge Total Pressure (pascal)	0	constant 💌
Supersonic/Initial Gauge Pressure (pascal)	0	constant 🔻
Direction Specification Method	Direction Vector	
Coordinate System	Cartesian (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	ntensity and Viscosity Ratio	•
	Turbulent Intensity (%) 5 P
	Turbulent Viscosity Rati	
ОК	Cancel Help	

- a. Select Direction Vector from the Direction Specification Method drop-down list.
- b. Enter 0 for X-Component of Flow Direction.

- c. Enter -1 for **Z-Component of Flow Direction**.
- d. Retain the selection of **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list.
- e. Retain the default, 5% for **Turbulence Intensity** and enter 5 for **Turbulent Viscosity Ratio**.
- f. Click **OK** to close the **Pressure Inlet** dialog box.
- 4. Retain the default settings for the pressure inlet of the stator (pressure-inlet-stator).

$lacksymbol{\bigcirc}$ Boundary Conditions $ ightarrow$	$\rightarrow \equiv$ pressure-inlet-stator \rightarrow	Edit
---	--	------

The profiles computed at the rotor outlet are used to update the boundary conditions at the stator inlet. These profiles were set automatically when the mixing plane was created. Therefore, you do not need to set any parameters in this dialog box.

Pressure Inlet	—				
Zone Name					
pressure-inlet-stator					
Momentum Thermal Radiation Species DPM Multiphase UDS	1				
Reference Frame Absolute	•				
Gauge Total Pressure (pascal)	or p0 👻				
Supersonic/Initial Gauge Pressure (pascal) pressure-outlet-rot	or p 🔻				
Direction Specification Method Direction Vector	•				
Coordinate System Cylindrical (Radial, Tangential, Axial)					
Radial-Component of Flow Direction pressure-outlet-rot	or ni 🔻				
Tangential-Component of Flow Direction pressure-outlet-rot	or nj 👻				
Axial-Component of Flow Direction pressure-outlet-rot	or nk 👻				
Turbulence					
Specification Method K and Epsilon	-				
Turbulent Kinetic Energy (m2/s2) pressure-outlet-rotor	r k 🔻				
Turbulent Dissipation Rate (m2/s3) pressure-outlet-rotor	re 🔻				
OK Cancel Help					

- a. Verify that the settings are defined by the fields of the **pressure-outlet-rotor** profile.
- b. Click **OK** to close the **Pressure Inlet** dialog box.
- 5. Retain the default settings for the pressure outlet of the rotor (pressure-outlet-rotor).

$\clubsuit Boundary \ Conditions \rightarrow \fbox pressure-outlet-rotor \rightarrow Edit...$

The **Backflow Direction Specification Method** was set to **Direction Vector** when you created the mixing plane, and the **Coordinate System** to **Cylindrical** (as for the stator inlet). The values for the direction cosines are taken from the profiles at the stator.

Pressure Outlet	
Zone Name	
pressure-outlet-rotor	
Momentum Thermal Radiation Species DPM Multiphase UDS	1
Gauge Pressure (pascal)	pressure-inlet-stator p 🔹
Backflow Direction Specification Method Direction Vector	•
Coordinate System Cylindrical (Radial, Tange	ential, Axial) 🔹
Radial-Component of Flow Direction	pressure-inlet-stator ni 🔹
Tangential-Component of Flow Direction	pressure-inlet-stator nj 🔹
Axial-Component of Flow Direction	pressure-inlet-stator nk 🔹
Radial Equilibrium Pressure Distribution	
Turbulence	
Specification Method K and Epsilon	•
Backflow Turbulent Kinetic Energy (m2/s2)	pressure-inlet-stator k
Backflow Turbulent Dissipation Rate (m2/s3)	pressure-inlet-stator e 🔹
OK Cancel Help	

- a. Verify that the settings are defined by the fields of the **pressure-inlet-stator** profile.
- b. Click **OK** to close the **Pressure Outlet** dialog box.
- 6. Set the conditions for the pressure outlet of the stator (pressure-outlet-stator).

 $\textcircled{P} Boundary Conditions \rightarrow \fbox{Pressure-outlet-stator} \rightarrow \texttt{Edit...}$

Pressure Outlet
Zone Name
pressure-outlet-stator
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method Normal to Boundary
Radial Equilibrium Pressure Distribution
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Viscosity Ratio
Backflow Turbulent Intensity (%)
Backflow Turbulent Viscosity Ratio
OK Cancel Help

a. Retain the default selection of **Normal to Boundary** from the **Backflow Direction Specification Method** drop-down list.

In problems where a backflow exists at the pressure outlet boundary (for example, a torque-converter), you can use this option to specify the direction of the backflow.

b. Enable the Radial Equilibrium Pressure Distribution option.

This option accounts for the pressure distribution that results from rotation by calculating the pressure gradient according to

$$\frac{\partial p}{\partial r} = \frac{\rho v_{\theta}^2}{r}$$

where v_{θ} is the tangential velocity. This option is appropriate for axial flow configurations with relatively straight flow paths (that is, little change in radius from inlet to exit).

- c. Retain the default selection of **Intensity and Viscosity Ratio** from the **Specification Method** dropdown list.
- d. Enter 1% for **Backflow Turbulent Intensity**.
- e. Enter 1 for Backflow Turbulent Viscosity Ratio.
- f. Click **OK** to close the **Pressure Outlet** dialog box.
- 7. Retain the default conditions for the **rotor-hub**.

```
\textcircled{}^{\bullet} Boundary \ Conditions \rightarrow \fbox{}^{\bullet} rotor-hub \rightarrow Edit...
```

For a rotating reference frame, ANSYS Fluent assumes by default that walls rotate with the rotating reference frame, and hence are stationary with respect to it. Since the **rotor-hub** is rotating, you should retain the default settings.

💶 Wall 💽
Zone Name
rotor-hub
Adjacent Cell Zone
fluid-rotor
Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film
Wall Motion Motion
Stationary Wall Noving Wall
Shear Condition
 No Slip Specified Shear Specularity Coefficient Marangoni Stress
Wall Roughness
Roughness Height (m) 0 constant -
Roughness Constant 0.5
OK Cancel Help

- a. Verify that Stationary Wall is selected from the Wall Motion list.
- b. Click **OK** to accept the default settings and close the **Wall** dialog box.
- 8. Set the conditions for the inlet hub of the rotor (rotor-inlet-hub).

 $\clubsuit Boundary \ Conditions \rightarrow \overleftarrow{\sqsubseteq} rotor-inlet-hub \rightarrow Edit...$

ne Name							
otor-inlet-hub							
ljacent Cell Zone							
luid-rotor							
Momentum Thermal	Radiation Species DPM M	tultiphase UDS 1	Vall Film				
Wall Motion	Motion						
C Stationary Wall	Relative to Adjacent Cell	700e				Speed (rpm)	
	 Absolute 					0	P
	Translational	Rotation-Axis	Origin		Rota	ation-Axis Direction	
	Rotational	X (m) 0		P	X	0	P
	Components	Y (m) 0			Y	0	
				P		•	P
		Z (m) 0		P	z	-1	P
Shear Condition							
 No Slip Specified Shear 							
O Specularity Coeffic	tient						
Marangoni Stress							
Wall Roughness							
Roughness Height (m	0 con	stant					
Roughness Constan	0.5 con	stant	-				

a. Select Moving Wall from the Wall Motion list.

The **Wall** dialog box will expand to show the wall motion inputs.

- b. Select Absolute and Rotational in the Motion group box.
- c. Enter -1 for **Z** in the **Rotation-Axis Direction** group box.
- d. Click **OK** to close the **Wall** dialog box.

These conditions set the **rotor-inlet-hub** to be a stationary wall in the absolute frame.

9. Set the conditions for the shroud of the rotor inlet (rotor-inlet-shroud).

$\textcircled{}^{\bullet} Boundary \ Conditions \rightarrow \fbox{}^{\bullet} rotor-inlet-shroud \rightarrow Edit...$

Wall One Name rotor-inlet-shroud djacent Cell Zone					
fluid-rotor					
Momentum Thermal	Radiation Species DPM Multiph	ase UDS Wall Film			
	Motion				
 Stationary Wall Moving Wall 	 Relative to Adjacent Cell Zone Absolute]		Speed (rpm)	P
	Translational	Rotation-Axis Origin		Rotation-Axis Direction	
	Rotational Components	X (m) 0	P	X 0	P
		Y (m) 0	P	YO	P
		Z (m) 0	P	Z -1	P
Shear Condition No Slip Specified Shear Specularity Coeffic Marangoni Stress	ient				
Wall Roughness					
Roughness Height (m)	0 constant	*			
Roughness Constant	0.5 constant	Ŧ			
	OK	Cancel Help			

- a. Select Moving Wall from the Wall Motion list.
- b. Select Absolute and Rotational in the Motion group box.
- c. Enter -1 for **Z** in the **Rotation-Axis Direction** group box.
- d. Click **OK** to close the **Wall** dialog box.

These conditions will set the **rotor-inlet-shroud** to be a stationary wall in the absolute frame.

10. Set the conditions for the rotor shroud (rotor-shroud).

 $\textcircled{P} Boundary Conditions \rightarrow \overleftarrow{e} rotor-shroud \rightarrow Edit...$

ne Name					
otor-shroud					
jacent Cell Zone					
uid-rotor					
	Radiation Species DPM Mu	ltiphase UDS Wall Film			
Vall Motion	Motion				
Stationary Wall Moving Wall	Relative to Adjacent Cell Z	one		Speed (rpm)	
(e) Horning Hon	Absolute			0	P
	C Translational	Rotation-Axis Origin		Rotation-Axis Direction	
	Rotational	X (m) 0	P	X O	P
	Components	Y (m)	_	YO	
			μ		P
		Z (m) 0	P	Z -1	P
					0
hear Condition					
No Slip Specified Shear					
 Specularity Coeffi 	cient				
Marangoni Stress					
Vall Roughness					
Roughness Height (m) 0 const	tant 👻			
Roughness Constan					
	© 0.5 const	tant 👻			

- a. Select Moving Wall from the Wall Motion list.
- b. Select **Absolute** and **Rotational** in the **Motion** group box.
- c. Enter -1 for **Z** in **Rotation-Axis Direction** group box.
- d. Click **OK** to close the **Wall** dialog box.

These conditions will set the **rotor-shroud** to be a stationary wall in the absolute frame.

13.4.9. Solution

1. Set the solution parameters.



Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	Â
Least Squares Cell Based 💌	
Pressure	
Second Order 👻	
Momentum	Ξ
Second Order Upwind 👻	
Turbulent Kinetic Energy	
Power Law 🔻	
Turbulent Dissipation Rate	
Power Law 👻	-
Transient Formulation	
· · · · · · · · · · · · · · · · · · ·	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
High Order Term Relaxation Options	
Default	
Help	

- a. Select Coupled from the Scheme drop-down list in the Pressure-Velocity Coupling group box.
- b. Ensure that **Second Order Upwind** is selected from the **Momentum** drop-down list in the **Spatial Discretization** group box.
- c. Select **Power Law** from the **Turbulent Kinetic Energy** drop-down list.
- d. Select **Power Law** from the **Turbulent Dissipation Rate** drop-down list.
- e. Enable the **Pseudo Transient** option.
- 2. Set the solution controls.

Solution Controls

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	Â
0.2	
Momentum	
0.5	Ξ
Density	
1	
Body Forces	
1	
Turbulent Kinetic Energy	
0.5	÷
Default	
Equations	
Help	

a. Enter 0.2 for Pressure in the Pseudo Transient Explicit Relaxation Factors group box.

- b. Enter 0.5 for **Turbulent Kinetic Energy**.
- c. Enter 0.5 for Turbulent Dissipation Rate.

Scroll down to find the Turbulent Dissipation Rate number-entry box.

Note

For this problem, it was found that these under-relaxation factors worked well.

For tips on how to adjust the under-relaxation parameters for different situations, see Setting Under-Relaxation Factors in the User's Guide.

3. Enable the plotting of residuals during the calculation.

```
♦ Monitors → 🔄 Residuals → Edit...
```

Options	Equations				
Print to Console	Residual	Monitor C	Check Convergen	ce Absolute Criteria	*
V Plot	continuity			0.001	
Vindow 1 Curves Axes	x-velocity			0.001	E
Iterations to Plot	y-velocity	\checkmark		0.001	
1000	z-velocity	V		0.001	-
	Residual Values			Convergence Cr	iterion
1000	Normalize		Iterations	absolute	
	Scale				
	Compute Loc	al Scale			

- a. Ensure that the **Plot** option is enabled in the **Options** group box.
- b. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Enable the plotting of mass flow rate at the flow exit.

 $\clubsuit Monitors (Surface Monitors) \rightarrow Create...$

Surface Monitor		×
Name	Report Type	
surf-mon-1	Mass Flow Rate	•
Options	Field Variable	
Print to Console	Pressure	Ψ.
V Plot	Static Pressure	-
Window	Surfaces	
2 Curves Axes	interior-16	
	interior-3	
V Write	periodic-11	
File Name	periodic-22	
surf-mon-1.out	pressure-inlet-rotor	=
	pressure-inlet-stator	
X Axis	pressure-outlet-rotor	
Iteration 👻	pressure-outlet-stator	
Cat Data Europe	rotor-blade	
Get Data Every	rotor-hub	
1 Iteration V	rotor-inlet-hub	-
Average Over	rotor-inlet-shroud	· ·
	Highlight Surfaces	
1	New Surface 🔻	
	Save Output Parameter	
O	Cancel Help	

- a. Retain **surf-mon-1** for **Name**.
- b. Enable the Plot and Write options.
- c. Retain surf-mon-1.out for File Name.
- d. Select Mass Flow Rate from the Report Type drop-down list.
- e. Select pressure-outlet-stator from the Surfaces selection list.
- f. Click **OK** to close the **Surface Monitor** dialog box.
- 5. Initialize the flow field.
 - **Over Solution** Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
More Settings Initialize
Patch
Reset DPM Sources Reset Statistics
[]
Help

- a. Retain the default selection of Hybrid Initialization from the Initialization Methods list.
- b. Click Initialize.

Note

A warning is displayed in the console stating that the convergence tolerance of 1.000000e-06 has not been 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 User's Guide.

c. Click More Settings... to open the Hybrid Initialization dialog box.

💶 Hybrid Initialization 🛛 🗾
General Settings Turbulence Settings Species Settings
Number of Iterations 15
Explicit Under-Relaxation Factor
Scalar Equation-0
Scalar Equation-1
Reference Frame
 Relative to Cell Zone Absolute
Initialization Options
Use Specified Initial Pressure on Inlets Use External-Aero Favorable Settings Maintain Constant Velocity Magnitude
OK Cancel Help

- i. Increase the Number of Iterations to 15.
- ii. Click **OK** to close the **Hybrid Initialization** dialog box.
- d. Click Initialize once more.

Note

Click **OK** in the **Question** dialog box to discard the current data. After completing 15 iterations, the console displays a message 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.

6. Save the case file (fanstage.cas.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}$

7. Start the calculation by requesting 200 iterations.



. . . .

Run Calculation
Check Case Preview Mesh Motion
Pseudo Transient Options
Fluid Time Scale
Time Step Method Pseudo Time Step (s)
User Specified O.01 Automatic
Number of Iterations Reporting Interval 200 Profile Update Interval 1 Data File Quantities Calculate
Help

a. Select User Specified from the Time Step Method list.

Note

While the **Automatic** method is suitable for most cases, for this problem you will specify a pseudo time step that is larger than the default value, in order to accelerate the convergence. For information on how the pseudo time step is automatically set, see Automatic Pseudo Transient Time Step in the Theory Guide.

- b. Enter 0.01 s for **Pseudo Time Step**.
- c. Enter 200 for Number of iterations.
- d. Click Calculate.

The solution will converge after approximately 110 iterations, as shown in Figure 13.3: Mass Flow Rate History (p. 576). However, the residual history plot is only one indication of solution convergence.

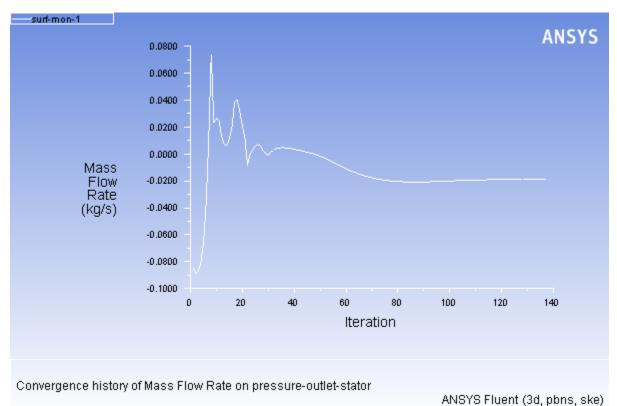


Figure 13.3: Mass Flow Rate History

8. Save the case and data files (fanstage-1.cas.gz and fanstage-1.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

9. Check the mass flux balance.

$\mathbf{O} \mathsf{Reports} \to \mathbf{Fluxes} \to \mathsf{Set} \mathsf{Up...}$

Warning

Although the mass flow rate history indicates that the solution is converged, you should also check the mass fluxes through the domain to ensure that mass is being conserved.

Flux Reports		—X
Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types axis exhaust-fan fan inlet-vent Boundary Name Pattern Save Output Parameter	Boundaries +interior- [2,0] +periodic- [2,0] +pressure- [4,4] +rotor- [5,0] +stator- [3,0]	Results 0.02541400492191315 0.01901420764625075 -0.0254138056188821 ≡ -0.0190138462930917 ▼ ▲ Ⅲ ▶ Net Results (kg/s) 5.606562e-07
Compute Write	Close He	alp

- a. Retain the default selection of Mass Flow Rate from the Options list.
- b. Click **(a)**, to the right of **Boundaries**, and select **+pressure-[4,0]** in the **Boundaries** group box.

This selects **pressure-inlet-rotor**, **pressure-inlet-stator**, **pressure-outlet-rotor**, and **pressure-outlet-stator**.

c. Click Compute.

Warning The net mass imbalance should be a small fraction (approximately 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.

Note

The fluxes are different for the portions of the rotor and stator that have been modeled. However, the flux for the whole rotor and the whole stator are both approximately equal to 0.2292 kg/s (that is, 0.02547 \times 9 rotor blades, and 0.01910 \times 12 stator blades).

d. Close the Flux Reports dialog box.

13.4.10. Postprocessing

1. Create an isosurface at *y*=0.12 m.

Surface \rightarrow Iso-Surface...

Iso-Surface		-x
Surface of Constant Mesh Y-Coordinate Min (m) 0.094457 Iso-Values (m)	From Surface interior-16 interior-3 periodic-11 periodic-22 pressure-inlet-rotor pressure-inlet-stator	
0.12 New Surface Name y=0.12	From Zones fluid-rotor fluid-stator	
Create Compute Manage.	Close Help	

The surface y=0.12 m is a midspan slice through the mesh. This view is useful for looking at the bladeto-blade flow field.

- 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 0.12 m for Iso-Values.
- d. Enter y=0.12 for New Surface Name.
- e. Click **Create** to create the isosurface.
- 2. Create an isosurface at z = -0.1 m.

Surface \rightarrow Iso-Surface...

The surface z = -0. 1 m is an axial plane downstream of the stator. This will be used to plot circumferentially averaged profiles.

a. Select Mesh... and Z-Coordinate from the Surface of Constant drop-down lists.

- b. Click **Compute** to update the minimum and maximum values.
- c. Enter -0.1 m for **Iso-Values**.
- d. Enter z = -0.1 for New Surface Name.

Note

The default name that ANSYS Fluent displays in the **New Surface Name** field (that is, **z-coordinate-17**) indicates that this is surface number 17. This fact will be used later in the tutorial when you plot circumferential averages.

- e. Click **Create** to create the isosurface.
- f. Close the **Iso-Surface** dialog box.
- 3. Display velocity vectors on the midspan surface y=0.12 (Figure 13.4: Velocity Vectors on y=0.12 Near the Stator Blade (p. 580)).

Graphics and Anim	ations $\rightarrow \stackrel{\frown}{\equiv}$ Vectors \rightarrow Se	et Up
Vectors		×
Options	Vectors of	
Global Range	Velocity	-
Auto Range	Color by	
 Clip to Range Auto Scale 	Velocity	•
Draw Mesh	Velocity Magnitude	•
Style	Min Max	
arrow	0 0	
Scale Skip	Surfaces	
10 2	rotor-inlet-shroud	*
Verter Ortere	rotor-shroud	
Vector Options	stator-blade	
Custom Vectors	stator-hub	E
	stator-shroud v=0.12	
Surface Name Pattern	y=0.12	*
Match	New Surface 🔻	
	Surface Types	
	axis	*
	clip-surf	
	exhaust-fan	
	fan	-
Display	Compute Close Help	

- a. Retain the default selection of arrow from the Style drop-down list.
- b. Enter 10 for Scale.

- c. Set Skip to 2.
- d. Select **y=0.12** from the **Surfaces** selection list.
- e. Click **Display** to plot the velocity vectors.
- f. Rotate and zoom the view to get the display shown in Figure 13.4: Velocity Vectors on y=0.12 Near the Stator Blade (p. 580).

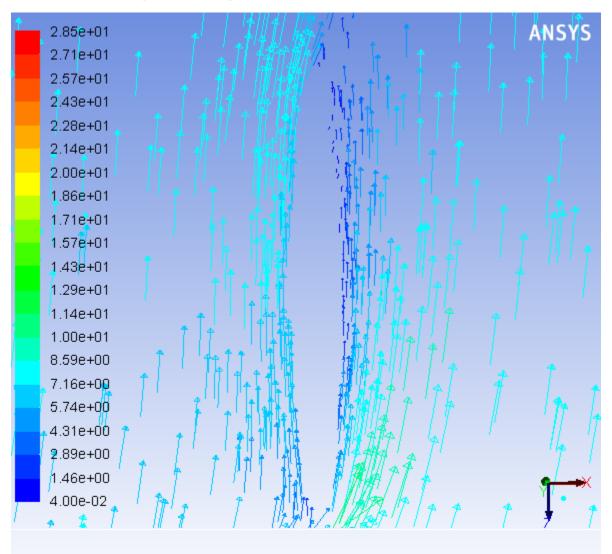


Figure 13.4: Velocity Vectors on y=0.12 Near the Stator Blade

Velocity Vectors Colored By Velocity Magnitude (m/s)

ANSYS Fluent (3d, pbns, ske)

Plotting the velocity field in this manner gives a good indication of the midspan flow over the stator. For the rotor, it is instructive to similarly plot the relative velocity field.

- g. Close the Vectors dialog box.
- 4. Plot a circumferential average of the total pressure on the plane z = -0.1.
 - a. Type the text commands in the console as follows:

```
>plot
/plot> circum-avg-radial
averages of> total-pressure
on surface [] 17
number of bands [5] 15
```

Note

Surface 17 is the surface **z=-0.1** you created earlier. For increased resolution, 15 bands are used instead of the default 5.

b. Enter the name of the output file as circum-plot.xy when prompted.

```
Computing r-coordinate ...
Clipping to r-coordinate ... done.
Computing "total-pressure" ...
Computing averages ... done.
Creating radial-bands surface (32 31 30 29 28 27 26 25 24 23 22 21 20 19 18).
filename [""] "circum-plot.xy"
```

c. Retain the default of no when asked to order points.

order points? [no] **no**

d. Display the circumferential average.



File XY Plot		×
Plot Title bircumferential Averages	Legend Title Total Pressure	
Files	Legend Entries	
C:\mixing_plane\solution_files\circum-plot.xy	Circumferential Averages	
		Add
		Delete
C:\mixing_plane\solution_files\circum-plot.xy	Circumferential Averages	Change Legend Entry
Plot Axes	Curves Close Help	

- i. Click Add... and select the file circum-plot.xy in the Select File dialog box.
- ii. Click **Plot** and close the **File XY Plot** dialog box.

The radial variation in the total pressure is very non-uniform, as shown in Figure 13.5: Plot of Circumferential Average of the Total Pressure on z=-0.1 Plane (p. 582). The losses are largest near the hub.

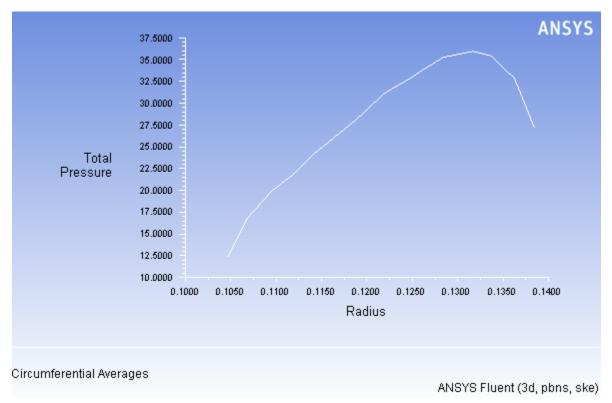


Figure 13.5: Plot of Circumferential Average of the Total Pressure on z=-0.1 Plane

5. Display filled contours of total pressure.

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

Contours	— ×
Options	Contours of
Filled	Pressure
 Node Values Global Range 	Static Pressure 🔹
Auto Range	Min (pascal) Max (pascal)
Clip to Range	-385.0939 279.8599
Draw Profiles Draw Mesh	Surfaces I I I
	pressure-outlet-stator
Levels Setup	radial-bands
	rotor-blade
	rotor-inlet-hub 👻
Surface Name Pattern	New Surface
Mate	Surface Types
	axis
	clip-surf
	exhaust-fan fan
	[an]
Display	Compute Close Help

- a. Enable **Filled** in the **Options** group box.
- b. Retain the default selections of **Pressure...** and **Total Pressure** from the **Contours of** drop-down lists.
- c. Select rotor-blade and rotor-hub from the Surfaces selection list.
- d. Click **Compute**.
- e. Click **Display** and close the **Contours** dialog box.
- f. Rotate the view to get the display as shown in Figure 13.6: Contours of Total Pressure for the Rotor Blade and Hub (p. 584).

The pressure contours are displayed in Figure 13.6: Contours of Total Pressure for the Rotor Blade and Hub (p. 584). Notice the high pressure that occurs on the leading edge of the rotor blade due to the motion of the blade.

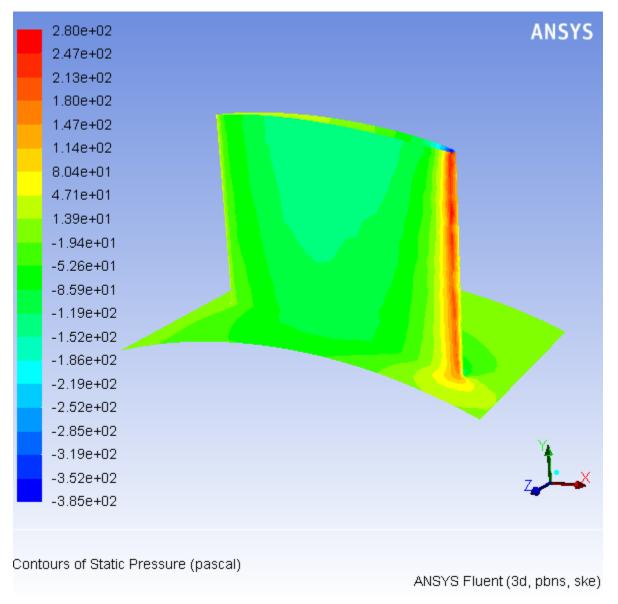


Figure 13.6: Contours of Total Pressure for the Rotor Blade and Hub

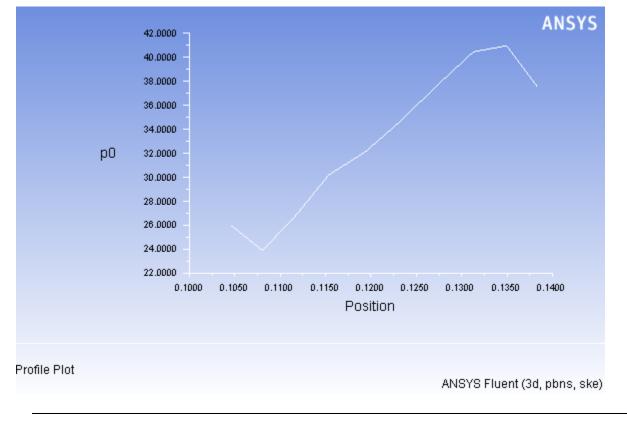
6. Display the total pressure profiles at the outlet of the rotor.

 $\diamondsuit \mathsf{Plots} \to \overleftarrow{\sqsubseteq} \mathsf{Profile \ Data} \to \mathsf{Set \ Up...}$

Plot Profile Data		
Profile pressure-outlet-rotor pressure-inlet-stator	Y Axis Function p0 p ni nj nk ni2 nj2 nk2 k	X Axis Function
Plot	Axes Curves Close	Help

- a. Ensure that **pressure-outlet-rotor** is selected from the **Profile** selection list.
- b. Ensure that **p0** is selected from the **Y Axis Function** selection list.
- c. Click **Plot** and close the **Plot Profile Data** dialog box.





Note

The profiles shown are area-averaged profiles computed by the mixing plane model.

13.5. Summary

This tutorial has demonstrated the use of the mixing plane model for a typical axial flow turbomachine configuration. The mixing plane model is useful for predicting steady-state flow in a turbomachine stage, where local interaction effects (such as wake and shock wave interaction) are secondary. If local effects are important, then a transient, sliding mesh calculation is required.

13.6. 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. 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. 123).

Chapter 14: Using Sliding Meshes

This tutorial is divided into the following sections:

14.1. Introduction14.2. Prerequisites14.3. Problem Description14.4. Setup and Solution14.5. Summary14.6. Further Improvements

14.1. 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.

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. 123)

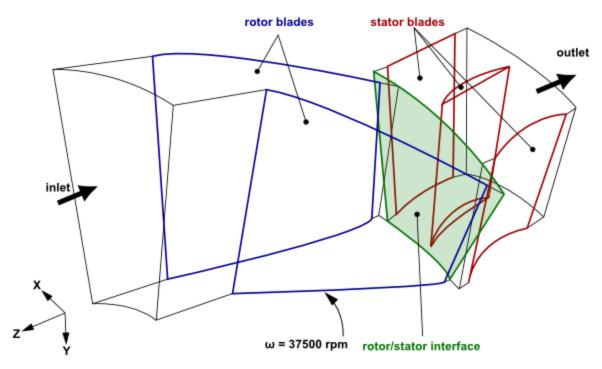
and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

14.3. Problem Description

The model represents a single-stage axial compressor comprised 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 comprised of only 13,856 cells total), the analysis will employ the inviscid model, so that ANSYS Fluent is solving the Euler equations.





14.4. Setup and Solution

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

14.4.1. Preparation
14.4.2. Mesh
14.4.3. General Settings
14.4.4. Models
14.4.5. Materials
14.4.6. Cell Zone Conditions
14.4.7. Boundary Conditions
14.4.8. Operating Conditions
14.4.9. Mesh Interfaces
14.4.10. Solution
14.4.11. Postprocessing

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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip sliding_mesh_R150.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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Enable single precision (disable **Double Precision**).
- 11. Run in Serial under Processing Options.

14.4.2. Mesh

1. Read in the mesh file axial_comp.msh.

 $\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

14.4.3. General Settings

1. Check the mesh.

General → Check

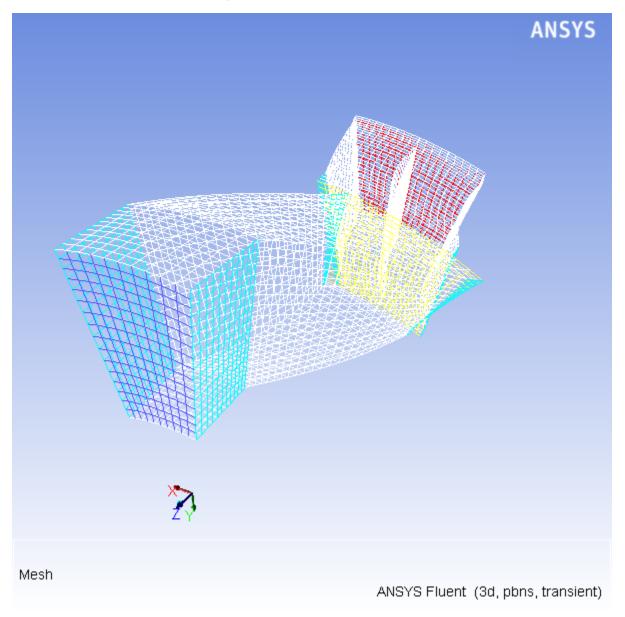
ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

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 14.2: Rotor-Stator Display (p. 590)).

Orient the view to display the mesh as shown in Figure 14.2: Rotor-Stator Display (p. 590). 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 14.2: 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
```

/mesh> modify-zones

/mesh/modify-zones> list-zones

id	name	type	material	kind
13	fluid-rotor	fluid	air	cell
28		fluid	air	cell
20	default-interior:0	interior	all	face
15		interior		face
15	rotor-hub	wall	aluminum	face
3 4				
-	rotor-shroud	wall	aluminum	face
7	rotor-blade-1	wall	aluminum	face
8	rotor-blade-2	wall	aluminum	face
16		wall	aluminum	face
17	stator-shroud	wall	aluminum	face
20	stator-blade-1	wall	aluminum	face
21	stator-blade-2	wall	aluminum	face
22	stator-blade-3	wall	aluminum	face
23	stator-blade-4	wall	aluminum	face
5	rotor-inlet	pressure-inlet		face
19	stator-outlet	pressure-outlet		face
10	rotor-per-1	wall	aluminum	face
12	rotor-per-2	wall	aluminum	face
24	stator-per-2	wall	aluminum	face
26	stator-per-1	wall	aluminum	face
6	rotor-interface	interface		face
18	stator-interface	interface		face
11	rotor-per-4	wall	aluminum	face
9	rotor-per-3	wall	aluminum	face
25	stator-per-4	wall	aluminum	face
27	stator-per-3	wall	aluminum	face
<u> </u>				

```
/mesh/modify-zones> make-periodic
Periodic zone [()] 10
Shadow zone [()] 9
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

General
Mesh
Scale Check Report Quality
Display
Solver
Type Velocity Formulation Pressure-Based Absolute
O Density-Based
Time Steady
 Transient
Gravity Units
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.

🗘 General	\rightarrow	Units

💶 Set Units			×
Quantities		Units	Set All to
acceleration angle angular-velocity		rad/s deg/s rpm	default
area area-inverse collision-rate concentration contact-resistance crank-angle crank-angular-velocity density density-gradient	~	Factor 0.1047198 Offset 0	si british cgs
[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.

14.4.4. Models

1. Enable the inviscid model.

```
\textcircled{Models} \rightarrow \overleftarrow{E} Viscous \rightarrow Edit...
```

💶 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)
O Detached Eddy Simulation (DES)
Carge Eddy Simulation (LES)
OK Cancel Help

- a. Select Inviscid in the Model list.
- b. Click **OK** to close the **Viscous Model** dialog box.

14.4.5. Materials

1. Specify air (the default material) as the fluid material, using the ideal gas law to compute density.



Create/Edit Materials		×
Name air	Material Type	Order Materials by
Chemical Formula	FLUENT Fluid Materials	Chemical Formula
	air	FLUENT Database User-Defined Database
	Mixture none	v
Properties		
Density (kg/m3)	ideal-gas 👻 Edit	
Cp (Specific Heat) (j/kg-k)	constant Edit Edit 1006.43	
Molecular Weight (kg/kgmol)	constant Edit 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.

14.4.6. Cell Zone Conditions

Cell Zone Conditions

Cell Zone Conditions			
Zone			
fluid-rotor fluid-stator			
Phase Type ID mixture Type ID			
Edit Copy Profiles Parameters Operating Conditions Display Mesh Porous Formulation Superficial Velocity Physical Velocity			
Help			

1. Set the boundary conditions for the fluid in the rotor (**fluid-rotor**).

 $\textcircled{Cell Zone Conditions} \rightarrow \fbox{fluid-rotor} \rightarrow \texttt{Edit...}$

Iuid			×
Zone Name			
fluid-rotor			
Material Name air			
Frame Motion Source Terms Mesh Motion Fixed Values			
Porous Zone			
Reference Frame Mesh Motion Porous Zone Embedded LE	S Reaction Sou	urce Terms Fixed Values Multiphase	
Relative Specification UDF			
	Motion Function	none 🔻	
Rotation-Axis Origin R	otation-Axis Direc	tion	
X (m) 0 constant -	X 0	constant 👻	
Y (m) 0 constant	YO	constant 🗸	
Z (m) 0 constant	z 1	constant 🗸	
	-		
Rotational Velocity	Translational Vel	ocity	
Speed (rpm) 37500 constant	X (m/s) 0	constant 👻	
Copy To Frame Motion	Y (m/s) 0	constant 👻	
	Z (m/s) 0	constant 🗸	
]			-
OK Ca	ncel 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 boundary conditions for the fluid in the stator (fluid-stator).

Cell Zone Conditions → $\overline{\equiv}$ fluid-stator → Edit...

Pluid	
Zone Name	
fluid-stator	
Material Name air 🗸 Edit]
Frame Motion Source Terms	
Mesh Motion Fixed Values	
Porous Zone	
Reference Frame Mesh Motion Porous Zone Ember	dded LES Reaction Source Terms Fixed Values Multiphase
Rotation-Axis Origin	Rotation-Axis Direction
X (m) 0 constant	X 0 constant
Y (m) 0 constant -	Y 0 constant
Z (m) 0 constant -	Z 1 constant
	-
<u>P</u>	
ОК	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.

14.4.7. Boundary Conditions

Boundary Conditions

Zone		
default-interior		
default-interior:0		
rotor-blade-1		
rotor-blade-2		
rotor-hub rotor-inlet		
rotor-interface		
rotor-per-1		
rotor-per-2		Ξ
rotor-shroud		
stator-blade-1		
stator-blade-2		
stator-blade-3		
stator-blade-4		
stator-hub		
stator-interface		
stator-outlet		-
stator-per-1		
Phase	Type ID	
mixture	✓ pressure-inlet ▼ 5	
Edit	Copy Profiles	
Parameters		
Parameters	Operating Conditions	
Display Mesh	Periodic Conditions	
Highlight Zone		
Help		

1. Set the boundary conditions for the inlet (rotor-inlet).

 $\clubsuit Boundary \ Conditions \rightarrow \overleftarrow{\sqsubseteq} rotor-inlet \rightarrow Edit...$

Pressure Inlet	
Zone Name	
rotor-inlet	
Momentum Thermal Radiation Species DPM Multiphase	UDS
Reference Frame Absolute	•
Gauge Total Pressure (atm)	constant 💌
Supersonic/Initial Gauge Pressure (atm) 0.9	constant 💌
Direction Specification Method Normal to Boundary	•
Non-Reflecting Boundary	
OK Cancel Help	

a. Enter 1.0 atm for Gauge Total Pressure.

- b. Enter 0.9 atm for Supersonic/Initial Gauge Pressure.
- c. Click the Thermal tab and enter 288 K for Total Temperature.

Pressure Inlet	×
Zone Name	
rotor-inlet	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Total Temperature (k) 288 constant	
OK Cancel Help	

- d. Click **OK** to close the **Pressure Inlet** dialog box.
- 2. Set the boundary conditions for the outlet (stator-outlet).

Boundary Conditions ->	stator-outlet → Edit
------------------------	----------------------

Pressure Outlet	×
Zone Name	
stator-outlet	
Momentum Thermal Radiation Species DPM Multiphase UDS	_
Gauge Pressure (atm) 1.08 constant	•
Backflow Direction Specification Method Normal to Boundary	•
Radial Equilibrium Pressure Distribution	
Target Mass Flow Rate	
Non-Reflecting Boundary	
OK Cancel 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 Outlet	×
Zone Name stator-outlet	
stator-oduet	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Backflow Total Temperature (k) 288 Constant	
OK Cancel 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.

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{\equiv}$ rotor-blade-1 \rightarrow Edit...

💶 Wall		X
Zone Name		
rotor-blade-	1	
Adjacent Cell	Zone	
fluid-rotor		
Momentum	Thermal Radiation Species DPM Multiphase	UDS Wall Film
This page is	not applicable under current settings.	
	OK Cancel 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.

14.4.8. Operating Conditions

1. Set the operating pressure.

Boundary Conditions -> Operating Conditions...

Operating Conditions	—
Pressure	Gravity
Floating Operating Pressure Operating Pressure (atm)	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 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.

14.4.9. Mesh Interfaces

1. Create a periodic mesh interface between the rotor and stator mesh regions.

Mesh Interfaces → Create/Edit...

Create/Edit Mesh Interfaces		×
Mesh Interface	Interface Zone 1	Interface Zone 2
int	rotor-interface	stator-interface
	rotor-interface	rotor-interface
	stator-interface	stator-interface
Interface Options	Boundary Zone 1	Interface Wall Zone 1
Periodic Boundary Condition Periodic Repeats		
Coupled Wall	Boundary Zone 2	Interface Wall Zone 2
Matching		
		Interface Interior Zone
Periodic Boundary Condition		
Type Offset		
Translational X (m) Rotational	Y (m) 0 Z (m) 0	
☑ Auto Compute Offset		
Create	Delete Draw List Close	Help

- 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 Zone 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 Zone 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.

 \bigcirc General \rightarrow Check

14.4.10. Solution

1. Set the solution parameters.

Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	Â
Least Squares Cell Based 🔹	
Pressure	
Second Order 🔹	
Density	Ξ
Second Order Upwind	
Momentum	
Second Order Upwind	
Energy	
Second Order Upwind 🔻	-
Transient Formulation	
First Order Implicit 🔹	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
High Order Term Relaxation Options	
Default	
Help	

a. Select **Coupled** from the **Pressure-Velocity Coupling** group box.

2. Change the 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	
Energy	
1	
Temperature	
0.9	
J	
	Ŧ
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.

 $\textcircled{} Monitors \rightarrow \overleftarrow{\sqsubseteq} Residuals \rightarrow Edit...$

Residual Monitors					×
Options	Equations Residual	Monitor	Check Convergence	Relative Criteria	
V Plot	continuity			0.01	
Window 1 Curves Axes	x-velocity			0.01	E
Iterations to Plot	y-velocity			0.01	
1000	z-velocity	V		0.01	-
	Residual Values			Convergence C	riterion
Iterations to Store	Normalize		Iterations 5	relative	
	Compute Loca	al Scale			
OK Plot	Renormaliz	e (Cancel He	lp	

- 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).

Monitors (Surface Monitors) → Create...

Name	Depart Turne	
	Report Type	
surf-mon-1	Mass Flow Rate	
Options	Field Variable	
✓ Print to Console	Pressure	
V Plot	Static Pressure	
Window	Surfaces	
2 Curves Axes	default-interior	
V Write	default-interior:0	
	interior-14	
File Name	rotor-blade-1	E
surf-mon-1.out	rotor-blade-2 rotor-hub	
X Axis	rotor-inlet	
Flow Time	rotor-interface	
	rotor-per-1	
Get Data Every	rotor-per-2	
1 Time Step 🔻	rotor-shroud	
Average Over(Time Steps)	stator-blade-1	
1	Highlight Surfaces	
-	New Surface	
ОК	Cancel Help	

- a. Retain the default entry of surf-mon-1 for Name.
- b. Enable **Plot** and **Write**.
- c. Retain the default entry of surf-mon-1.out for File Name.
- d. Select Flow Time from the X Axis drop-down list.
- e. Select Time Step from the Get Data Every drop-down list.
- f. Select Mass Flow Rate from the Report Type drop-down list.
- g. Select rotor-inlet from the Surfaces selection list.
- h. Click **OK** to close the **Surface Monitor** dialog box.
- 5. Enable the plotting of mass flow rate at the outlet (stator-outlet).

♦ Monitors (Surface Monitors) → Create...

Surface Monitor		×
Name	Report Type	
surf-mon-2	Mass Flow Rate	-
Options	Field Variable	
Print to Console	Pressure	Ψ
V Print to Console	Static Pressure	Ŧ
Window	Surfaces	
3 Curves Axes	rotor-shroud	*
	stator-blade-1	
Write	stator-blade-2	
File Name	stator-blade-3	
surf-mon-2.out	stator-blade-4	
	stator-hub	
X Axis	stator-interface	
Flow Time	stator-outlet	E
Cat Data Every	stator-per-1	-
Get Data Every	stator-per-2	
1 Time Step 🔻	stator-shroud	*
Average Over(Time Steps)	Highlight Surfaces	
1		
	New Surface 💌	
ОК	Cancel Help	

- a. Retain the default entry of **surf-mon-2** for **Name**.
- b. Enable **Plot** and **Write**.
- c. Retain the default entry of **surf-mon-2.out** for **File Name**.
- d. Select Flow Time from the X Axis drop-down list.
- e. Select Time Step from the Get Data Every drop-down list.
- f. Select Mass Flow Rate from the Report Type drop-down list.
- g. Select stator-outlet from the Surfaces selection list.
- h. Click **OK** to close the **Surface Monitor** dialog box.
- 6. Enable the plotting of the area-weighted average of the static pressure at the interface (stator-interface).

Monitors (Surface Monitors) → Create...

4 Curves Axes rotor-shroud stator-blade-1 stator-blade-2 stator-blade-3 stator-blade-4 stator-hub X Axis Flow Time Get Data Every 1 Time Step Average Over(Time Stepc)	💶 Surface Monitor	
Options Options Image: Print to Console Image: Pressure Image: Pressure <t< td=""><td>Name</td><td>Report Type</td></t<>	Name	Report Type
Options Pressure Pressure Pressure Static Pressure Surfaces Surfaces Surfaces rotor-shroud stator-blade-1 stator-blade-2 stator-blade-2 stator-blade-3 stator-blade-3 stator-blade-4 stator-blade-4 stator-blade-4 stator-outlet stator-outlet stator-outlet stator-per-1 stator-per-2 stator-shroud	surf-mon-3	Area-Weighted Average 🔹
Print to Console Plot Window 4 Curves Axes Surfaces Image: Surf-mon-3.out X Axis Flow Time Get Data Every Image: Surf-mon-2 Outer (Time Step) Pressure Static Pressure Surfaces Surfaces Image: Surface Surfaces	Ontions	Field Variable
Vindow 4 Curves 4 Curves Axes Write File Name surf-mon-3.out X Axis Flow Time Get Data Every 1 Time Step Static Pressure Surfaces Static Pressure Surfaces rotor-shroud stator-blade-1 stator-blade-2 stator-blade-3 stator-hub stator-interface stator-outlet stator-per-1 stator-per-2 stator-shroud		Pressure 🔻
4 Curves Axes rotor-shroud stator-blade-1 stator-blade-2 stator-blade-3 stator-blade-4 stator-hub stator-hub stator-outlet stator-outlet stator-per-1 stator-per-2 stator-shroud 4 Curves Axes V Write Stator-blade-1 Stator-blade-3 Stator-hub X Axis Flow Time Get Data Every Time Step Ausrage Ouge(Time Stage) Ausrage Ouge(Time Stage) 		Static Pressure 💌
Write File Name surf-mon-3.out X Axis Flow Time Get Data Every 1 Time Step Ausrass Ourge(Time Stang)	Window	Surfaces
1 ▲ New Surface ▼	V Write File Name Surf-mon-3.out X Axis Flow Time Get Data Every 1 Time Step Average Over(Time Steps)	stator-blade-1 stator-blade-2 stator-blade-3 stator-blade-4 stator-hub stator-interface stator-outlet stator-per-1 stator-per-2 stator-shroud

- a. Retain the default entry of surf-mon-3 for Name.
- b. Enable **Plot** and **Write**.
- c. Retain the default entry of surf-mon-3.out for File Name.
- d. Select Flow Time from the X Axis drop-down list.
- e. Select Time Step from the Get Data Every drop-down list.
- f. Select Area-Weighted Average from the Report Type drop-down list.
- g. Retain the default selection of **Pressure...** and **Static Pressure** from the **Field Variable** drop-down lists.
- h. Select stator-interface from the Surfaces selection list.
- i. Click **OK** to close the **Surface Monitor** dialog box.
- 7. Initialize the solution using the values at the inlet (**rotor-inlet**).

Solution Initialization

Solution Initialization	
Initialization Methods	
 Hybrid Initialization Standard Initialization 	
Compute from	
rotor-inlet 🔹	
Reference Frame	
 Relative to Cell Zone Absolute 	
Initial Values	
Gauge Pressure (atm)	^
0.9000002	
X Velocity (m/s)	
2.518036e-05	
Y Velocity (m/s)	
1.394159e-06	
Z Velocity (m/s)	=
-130.9983	
Temperature (k)	
279.4745	
	Ŧ
Initialize Reset Patch	
Reset DPM Sources Reset Statistics	
Help	

- 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).

```
\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}
```

9. Run the calculation for one revolution of the rotor.

CRun Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method	Time Step Size (s)
Fixed 🔹	6.6667e-06
Settings	Number of Time Steps
Options	
Extrapolate Variables Data Sampling for Time S Sampling Interval	Sampling Options
Max Iterations/Time Step	Reporting Interval
Profile Update Interval	Acoustic Signals
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.

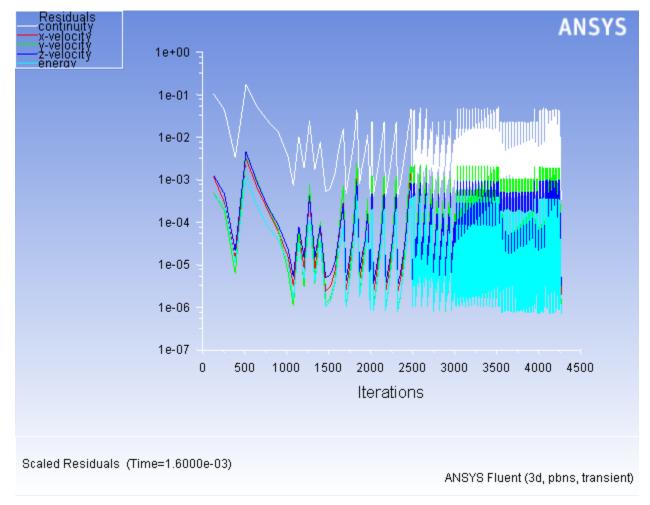


Figure 14.3: Residual History for the First Revolution of the Rotor

10. Examine the monitor histories for the first revolution of the rotor (Figure 14.4: Mass Flow Rate at the Inlet During the First Revolution (p. 612), Figure 14.5: Mass Flow Rate at the Outlet During the First Revolution (p. 613), and Figure 14.6: Static Pressure at the Interface During the First Revolution (p. 614)).

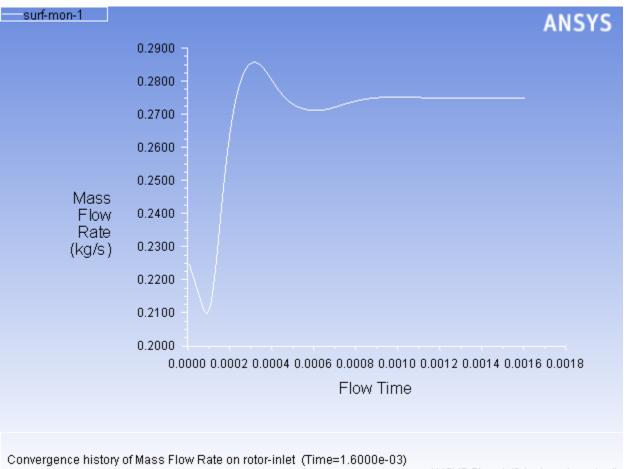


Figure 14.4: Mass Flow Rate at the Inlet During the First Revolution

ANSYS Fluent (3d, pbns, transient)

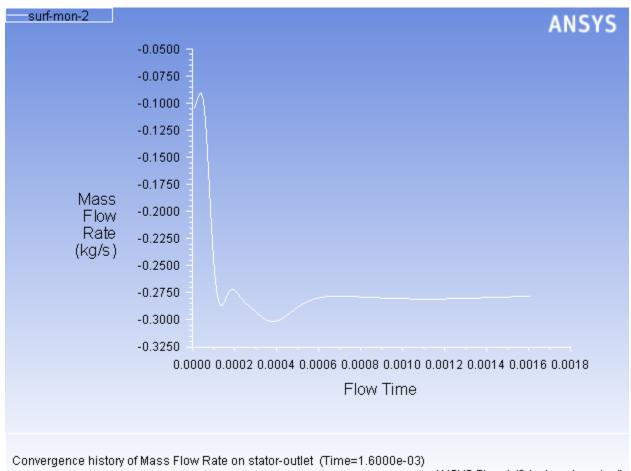
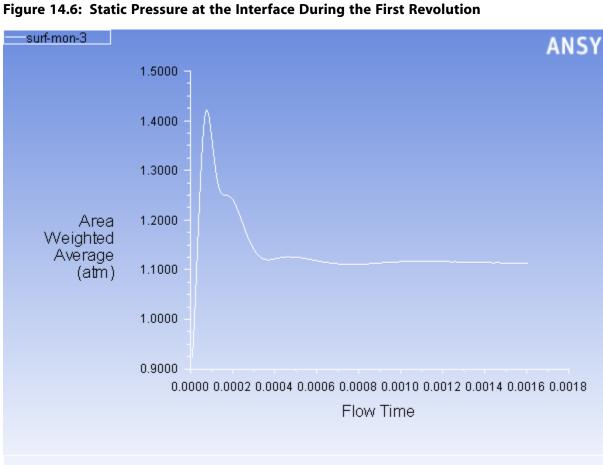


Figure 14.5: Mass Flow Rate at the Outlet During the First Revolution

ANSYS Fluent (3d, pbns, transient)



Convergence history of Static Pressure on stator-interface (Time=1.6000e-03)

ANSYS Fluent (3d, pbns, transient)

The monitor 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).

File \rightarrow Write \rightarrow Case & Data...

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 monitor files in preparation for further iterations.

$\clubsuit Monitors \rightarrow \blacksquare surf-mon-1 \rightarrow Edit...$

By saving the monitor histories under a new file name, the range of the axes will automatically be set to show only the data generated during the next set of iterations. This will scale the plots so that the fluctuations are more visible.

💶 Surface Monitor	×
Name	Report Type
surf-mon-1	Mass Flow Rate 🔹
Options	Field Variable
Print to Console	Pressure 🔻
V Plot	Static Pressure 👻
Window	Surfaces
2 Curves Axes	default-interior
V Write	default-interior:0
	interior-14
File Name	rotor-blade-1
surf-mon-1b.out	rotor-blade-2
X Axis	rotor-hub
	rotor-interface
Flow Time 🔻	rotor-per-1
Get Data Every	rotor-per-2
1 Time Step	rotor-shroud
	stator-blade-1
Average Over(Time Steps)	Highlight Surfaces
1	
	New Surface 🔻
OK	Cancel Help

- a. Enter surf-mon-1b.out for File Name.
- b. Click **OK** to close the **Surface Monitor** dialog box.
- 13. Similarly, rename surf-mon-2.out and surf-mon-3.out 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.



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	Chalifation
Data Sampling for Time Sampling Interval	Stausucs
	Sampling Options
Time Sampled (s) 0
Max Iterations/Time Step	Reporting Interval
20	
Profile Update Interval	
1	
Data File Quantities	Acoustic Signals
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 monitor histories for the next three revolutions of the rotor to verify that the solution is time-periodic (Figure 14.7: Mass Flow Rate at the Inlet During the Next 3 Revolutions (p. 617) Figure 14.8: Mass Flow Rate at the Outlet During the Next 3 Revolutions (p. 618), and Figure 14.9: Static Pressure at the Interface During the Next 3 Revolutions (p. 619)).

Note

If you read the provided data file instead of iterating the solution for three revolutions, the monitor histories can be displayed by using the **File XY Plot** dialog box.

\bigcirc Plots → **\overline{=}** File → Set Up...

Click the **Add** button in the **File XY Plot** dialog box to select one of the monitor 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 14.7: Mass Flow Rate at the Inlet During the Next 3 Revolutions

ANSYS Fluent (3d, pbns, transient)

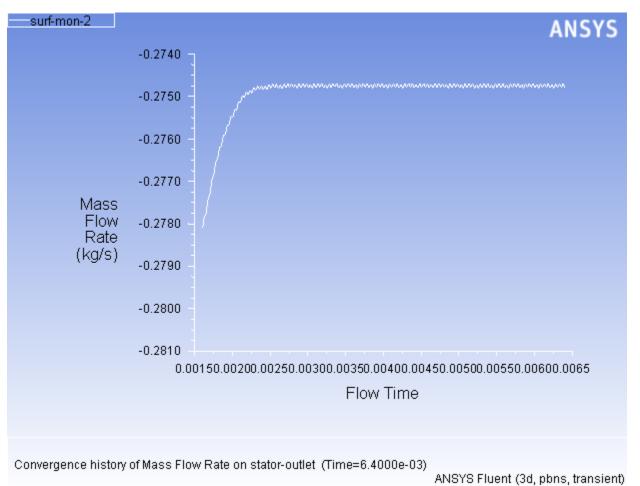


Figure 14.8: Mass Flow Rate at the Outlet During the Next 3 Revolutions

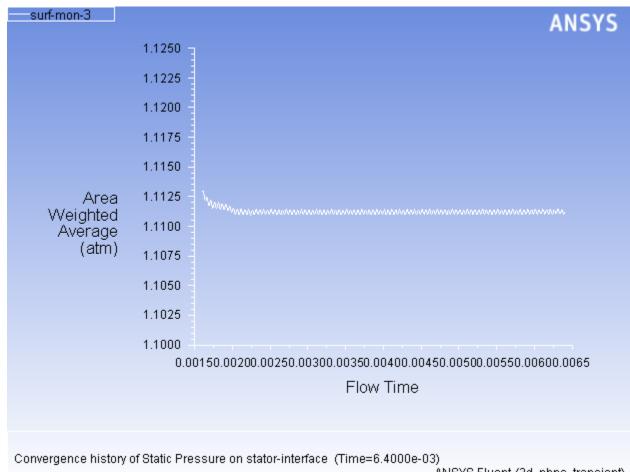


Figure 14.9: Static Pressure at the Interface During the Next 3 Revolutions

ANSYS Fluent (3d, pbns, transient)

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.

Surface \rightarrow Point...

💶 Point Surface	•••
Options	Coordinates
Point Tool	x0 (m) -0.02
Reset	y0 (m) -0.08
	z0 (m) -0.036
5	elect Point with Mouse
New Surface Name	2
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**).

Report Type
Vertex Average
Field Variable
Static Pressure
Surfaces
default-interior default-interior:0 interior-14
point-1
rotor-blade-1 rotor-blade-2 rotor-hub
rotor-inlet rotor-interface
rotor-per-1
rotor-per-2
rotor-shroud ☐ Highlight Surfaces New Surface ▼

$\clubsuit Monitors (Surface Monitors) \rightarrow Create...$

a. Retain the default entry of surf-mon-4 for Name.

- b. Enable **Plot** and **Write**.
- c. Retain the default entry of **surf-mon-4.out** for **File Name**.
- d. Select Flow Time from the X Axis drop-down list.
- e. Select Time Step from the Get Data Every drop-down list.
- f. Select Vertex Average from the Report Type drop-down list.
- g. Retain the defaults of Pressure and Static Pressure for Field Variable.
- h. Select **point-1** from the **Surfaces** selection list.
- i. Click OK to close the Surface Monitor dialog box.
- 20. Continue the calculation for one final revolution of the rotor, while saving data samples for the postprocessing of the time statistics.

Run Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method T Fixed	ime Step Size (s) 6.6667e-06
Settings	Aumber of Time Steps
Options	
Extrapolate Variables Data Sampling for Time St Sampling Interval 1 Sampled (s)	atistics ampling Options
Max Iterations/Time Step	1
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- a. Enter 240 for Number of Time Steps.
- b. Enable Data Sampling for Time Statistics in the Options group box.

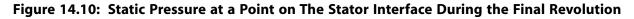
Enabling **Data Sampling for Time Statistics** causes ANSYS Fluent to calculate and store mean and root-mean-square (RMS) values of various quantities and field functions over the calculation interval.

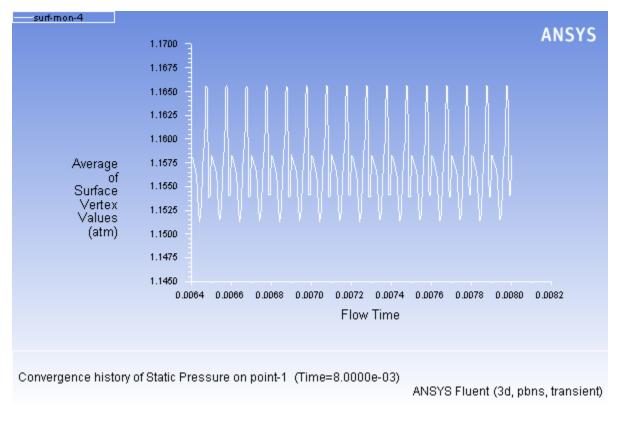
c. Click Calculate.

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...





14.4.11. 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).

♦ Plots
$$\rightarrow \stackrel{\bullet}{=}$$
 FFT \rightarrow Set Up...

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 Convergence history of Mass Flow Rate c
Y Axis Label	X Axis Label
Power Spectral Density	Frequency (Hz)
Files C:\Users\wescott\Documents\My Documents\Train	Load Input File Free File Data
C:\Users\wescott\Documents\My Documents\Tra	
Plot FFT Axes	Curves Close Help

a. Click the **Load Input File...** button to open the **Select File** dialog box.

Select File			×
Look in:	🎳 sliding_mesh 👻	G 🌶 📂 🖽 -	
(Pa)	Name	Date modified	Туре
	solution_files	4/22/2013 1:09 PM	File folder
Recent Places	axial_comp.msh	4/12/2013 9:21 AM	MSH File
	surf-mon-1.out	4/17/2013 9:35 AM	OUT File
	surf-mon-1b.out	4/17/2013 10:16 AM	OUT File
Desktop	surf-mon-1c.out	4/17/2013 10:44 AM	OUT File
	surf-mon-2.out	4/17/2013 9:35 AM	OUT File
678	surf-mon-2b.out	4/17/2013 10:16 AM	OUT File
Libraries	surf-mon-2c.out	4/17/2013 10:44 AM	OUT File
	surf-mon-3.out	4/17/2013 9:35 AM	OUT File
	surf-mon-3b.out	4/17/2013 10:16 AM	OUT File
Computer	surf-mon-3c.out	4/17/2013 10:44 AM	OUT File
Naturali	٠ III		÷.
Network	Input Signal File surf-mon-1c.out	•	ОК
	Files of type: All Files	•	Cancel

- i. Select **All Files** from the **files of type:** drop-down list.
- ii. Select surf-mon-lc.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	Signal Statistics	Window
Clip to Range	Min	None 👻
Subtract Mean Value Subdivide into Segments	0.2747448	Segment Control
	Max	Control Method
X Axis Range	0.2747599	Samples
Min 0.006420032	Mean	Frequency
	0.2747526	Samples per Segment
Max 0.00800004	Variance	238
0.00800004	2.267885e-11	Frequency Resolution (hz)
Set Defaults	Number of Samples	632.8918
Serberauts	238	Overlap (0 to 1)
	Min Frequency (hz)	0
	632.9081	
Signal Plot Title		
Convergence history of Mass Flow Rate on rotor-inlet (in SI units)		
Y Axis Label	X Axis Label	
Mass Flow Rate Flow Time		
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.

$\mathbf{O} \mathsf{Plots} \to \mathbf{E} \mathsf{FFT} \to \mathsf{Set} \mathsf{Up...}$

- 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.

Options	Signal Statistics	Window
Clip to Range	Min	None 🔻
Subtract Mean Value Subdivide into Segments	-0.2748007	Segment Control
	Max	Control Method
Axis Range	-0.2746998	 Samples Frequency
Min 0.006420032	Mean	Frequency
Max	-0.2747511	Samples per Segment
0.00800004	Variance	238
0.0000004	9.179356e-10	Frequency Resolution (hz)
Set Defaults	Number of Samples	632.8918
Serberbans	238	Overlap (0 to 1)
	Min Frequency (hz)	0
	632.9081	
Signal Plot Title		
Convergence history of Mass Flow Rate on stator-outlet (in SI units)		
Axis Label	X Axis Label	
Mass Flow Rate	Flow Time	

- 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 14.11: Area-Weighted Average Mass Flow Rate at the Outlet During the Final Revolution (p. 626)).

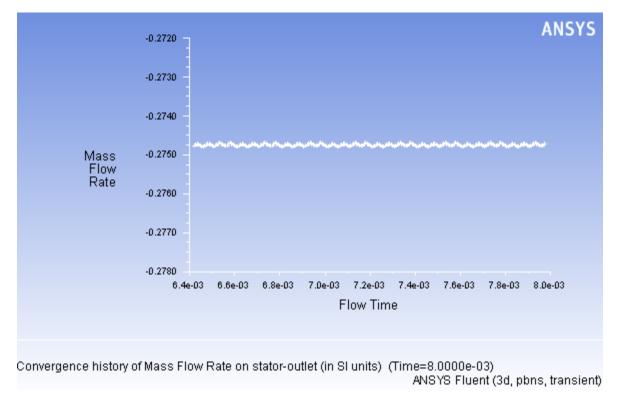


Figure 14.11: Area-Weighted Average Mass Flow Rate at the Outlet During the Final Revolution

- 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.

♦ Plots $\rightarrow \overline{\equiv}$ FFT \rightarrow Set Up...

- 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.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.

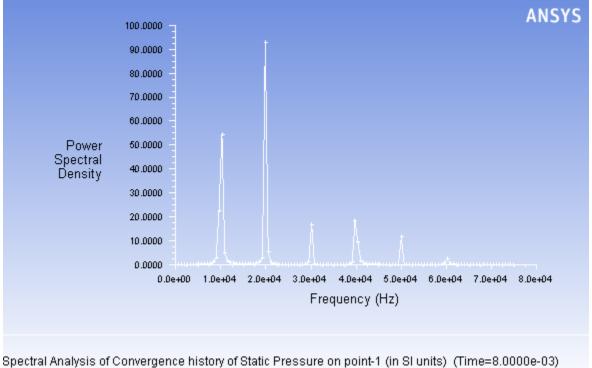
Options	Signal Statistics	Window
Clip to Range	Min	None 👻
Subtract Mean Value Subdivide into Segments	116657.2	Segment Control
	Max	Control Method
Axis Range	118108.1	Samples
Min	Mean	Frequency
0.006420032	117231.8	Samples per Segment
Max	Variance	238
0.00800004	172228.7	Frequency Resolution (hz)
Set Defaults	Number of Samples	632.8918
	238	Overlap (0 to 1)
	Min Frequency (hz)	0
	632.9081	
ignal Plot Title		
Convergence history of Static	Pressure on point-1 (in Si	(units)
Axis Label	X Axis Label	
Vertex Average Static Pressu		

- 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		
Axis a X Y Label	Number Format Type exponential Precision 1 Type Precision Type	Major Rules Color foreground Weight 1
Options Log Auto Range Major Rules Minor Rules	Range Minimum 0 Maximum 0	Minor Rules Color dark gray Weight 1
	Apply Close Help	

- 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 14.12: FFT of Static Pressure at the Stator



ANSYS Fluent (3d, pbns, transient)

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.

_	
Contours	
Options	Contours of
V Filled	Unsteady Statistics 👻
 Node Values Global Range 	Mean Static Pressure
Auto Range	Min Max
Clip to Range	0
Draw Profiles	
Draw Mesh	Surfaces
	stator-interface
Levels Setup	stator-outlet
	stator-per-1
20 🔺 1 🛋	stator-per-2
	stator-shroud
Surface Name Pattern	New Surface 🔻
Match	
	Surface Types
	symmetry 🔺
	velocity-inlet
	wall
	zone-surf 👻
Display Compute Close Help	

\bigcirc Graphics and Animations → **\equiv** Contours → Set Up...

- 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 Surface Types 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 14.13: Mean Static Pressure on the Outer Shroud of the Axial Compressor (p. 630).

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.

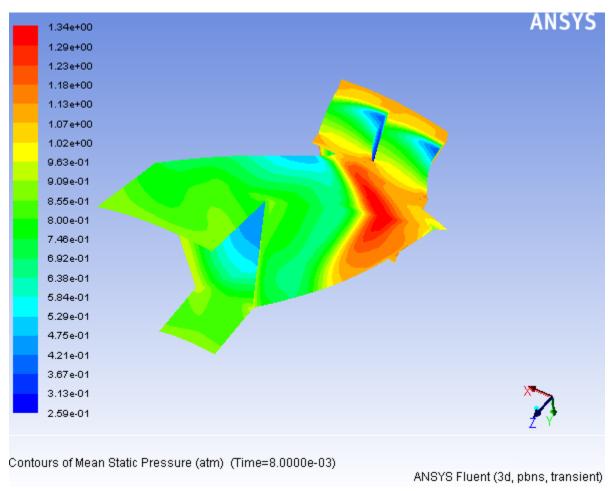


Figure 14.13: Mean Static Pressure on the Outer Shroud of the Axial Compressor

14.5. 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 monitored 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 monitor data. You also used the FFT utility to examine the frequency content of the transient monitor data. The observed peak corresponds to the passing frequency and the higher harmonics of the passing frequency, which occurred at approximately 10,000 Hz.

14.6. 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. 123).

Chapter 15: Using Dynamic Meshes

This tutorial is divided into the following sections:

15.1. Introduction
15.2. Prerequisites
15.3. Problem Description
15.4. Setup and Solution
15.5. Summary
15.6. Further Improvements

15.1. 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. In this tutorial, such a case will be used to demonstrate the layering feature of the dynamic mesh capability in ANSYS Fluent.

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, the check valve is connected to a spring that acts to push the valve to the valve seat and to shut the flow. But when the pressure force on the valve is greater than the spring force, the valve 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 valve is typically neglected, thus allowing for a rigid body Fluid Structure Interaction (FSI) calculation, for which a user-defined function (UDF) is provided.

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.
- · Compile a User-Defined Function (UDF) to specify flow-driven rigid-body motion.
- Calculate a solution using the pressure-based solver.

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)

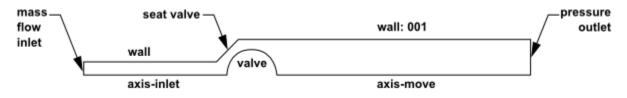
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

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

15.3. Problem Description

The check valve problem to be considered is shown schematically in Figure 15.1: Problem Specification (p. 632). A 2D axisymmetric valve geometry is used, consisting of a mass flow inlet on the left, and a pressure outlet on the right, driving the motion of a valve. In this case, the transient motion of the valve due to spring force, gravity, and hydrodynamic force is studied. Note, however, that the valve in this case is not completely closed. Since dynamic mesh problems require that at least one layer remains in order to maintain the topology, a small gap will be created between the valve and the valve seat.

Figure 15.1: Problem Specification



15.4. Setup and Solution

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

15.4.1. Preparation
15.4.2. Mesh
15.4.3. General Settings
15.4.4. Models
15.4.5. Materials
15.4.6. Boundary Conditions
15.4.7. Solution: Steady Flow
15.4.8. Time-Dependent Solution Setup
15.4.9. Mesh Motion
15.4.10. Time-Dependent Solution
15.4.11. Postprocessing

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 15.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip dynamic_mesh_R150.zip to your working folder.

The mesh and source files valve.msh and valve.c can be found in the dynamic_mesh directory created after unzipping the file.

A user-defined function will be used to define the rigid-body motion of the valve geometry. This function has already been written (valve.c). You will only need to compile it within ANSYS Fluent.

8. Use the Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

Note that this tutorial has been generated using single precision, so you should ensure that **Double Precision** is disabled if you want to match the tutorial setup exactly.

For more information about Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Run in Serial under Processing Options.

15.4.2. Mesh

1. Read the mesh file valve.msh.

 $\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

15.4.3. General Settings

1. Check the mesh.



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 mm.

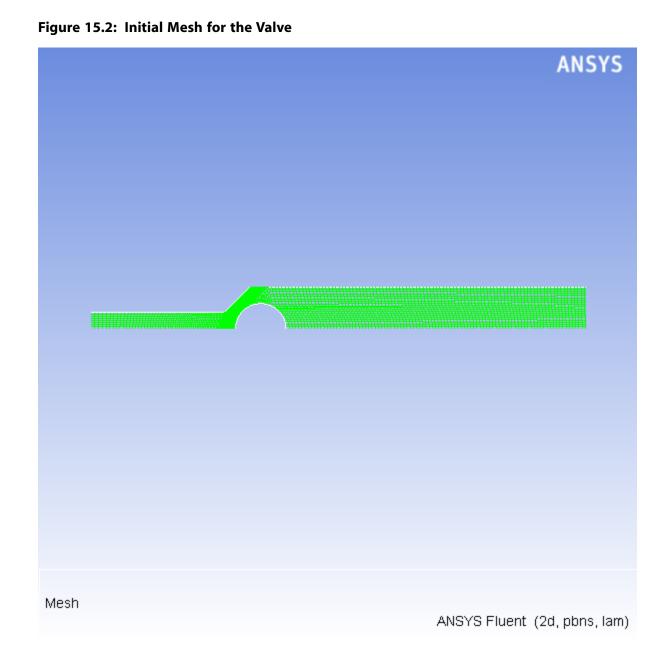
♀General → Units...

- a. In the Set Units dialog box select length under Quantities and mm under Units.
- b. Close the **Set Units** dialog box.
- 3. Display the mesh (Figure 15.2: Initial Mesh for the Valve (p. 635)).

$\clubsuit General \rightarrow Display...$

💶 Mesh Displa	у		—
Options Nodes Edges Faces Partitions	Edge Type a All Feature Outline	Surfaces axis-inlet axis-move default-interior default-interior:012 inlet	
0	Feature Angle	int-layering outlet	-
Outline Inter	Match	New Surface Surface Types axis clip-surf exhaust-fan fan	
Display Colors Close Help			

- a. Deselect axis-inlet, axis-move, inlet, and outlet from the Surfaces selection list.
- b. Click **Display**.



- c. Close the **Mesh Display** dialog box.
- 4. Enable an axisymmetric steady-state calculation.



General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative
Time	2D Space
 Steady Transient 	 Axisymmetric
- Hanalent	Axisymmetric Swirl
Gravity	Units
Help	

a. Select Axisymmetric from the 2D Space list.

15.4.4. Models

*					
4 1					
Ŷ	м	n	d	ρ	I٩
		~	~	-	

Models
Models
Multiphase - Off
Energy - Off
Viscous - Laminar
Radiation - Off
Heat Exchanger - Off
Species - Off Discrete Phase - Off
Solidification & Melting - Off
Acoustics - Off
Edit
Lucin
Help

1. Enable the standard k- ε turbulence model.



Viscous Model	
Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) k-epsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Scalable Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment User-Defined Wall Functions Enhanced Wall Treatment Pressure Gradient Effects Options Production Kato-Launder Production Limiter	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number IDR Prandtl Number IDR Prandtl Number Inone
OK	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.

15.4.5. Materials

Materials

Materials

Materials
Fluid
air
Solid aluminum
Create/Edit
Create/Edit Delete
Help

1. Apply the ideal gas law for the incoming air stream.

```
 \diamondsuit \mathsf{Materials} \to \mathbf{Fluid} \to \mathsf{Create/Edit...}
```

1		Order Materials by
lame	Material Type	Order Materials by
air	fluid	O Name O Chemical Formula
Chemical Formula	Fluent Fluid Materials	
	air	Fluent Database
	Mixture	User-Defined Database
	none	-
roperties		
Density (kg/m3)	ideal-gas 🗸 Edit	
Cp (Specific Heat) (j/kg-k)	constant 👻 Edit	
	1006.43	
Thermal Conductivity (w/m-k)		
mermai Conductivity (w/m-k)	constant v Edit	
	0.0242	
Viscosity (kg/m-s)		
viscosity (kg/m-s/	constant	
	1.7894e-05	
	· ·	

- a. Select ideal-gas from the Density drop-down list.
- b. Click Change/Create.
- c. Close the Create/Edit Materials dialog box.

15.4.6. 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.

Boundary	Conditions
Dogingary	vonutions

Zone		
axis-inlet axis-move default-interior default-interior:0	12	
inlet int-layering outlet seat-valve valve wall wall:001		
Phase mixture	Type ▼ mass-flow-inlet ▼	ID 11
Edit Parameters Display Mesh	Copy Profiles Operating Conditions Periodic Conditions	
Help		

- a. Select mass-flow-inlet from the Type drop-down list in the Boundary Conditions task page.
- b. Click Yes when ANSYS Fluent asks you if you want to change the zone type.

The *Mass-Flow Inlet* boundary condition dialog box will open.

Mass-Flow Inlet	×			
Zone Name				
inlet				
Momentum Thermal Radiation Species DPM Multiphase U	DS			
Reference Frame Absolute	•			
Mass Flow Specification Method Mass Flow Rate				
Mass Flow Rate (kg/s) 0.0116	constant 🔻			
Supersonic/Initial Gauge Pressure (pascal)	constant 🔹			
Direction Specification Method Normal to Boundary				
Turbulence				
Specification Method Intensity and Hydraulic Diam	eter 👻			
Turbulent Intensity (%	6) 5 P			
Hydraulic Diameter (mr	n) 20			
OK Cancel Help				

- i. Enter 0.0116 kg/s for Mass Flow Rate.
- ii. Select Normal to Boundary from the Direction Specification Method drop-down list.
- iii. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
- iv. Retain 5% for Turbulent Intensity.
- v. Enter 20 mm for the Hydraulic Diameter.
- vi. Click **OK** to close the **Mass-Flow Inlet** dialog box.
- 2. Set the conditions for the exit boundary (**outlet**).

$\clubsuit Boundary \ Conditions \rightarrow \overleftarrow{E} \ outlet$

Boundary Conditions

Zone		
axis-inlet axis-move default-interior default-interior:0: inlet int-layering outlet	12	
seat-valve valve wall wall:001		
Phase mixture	Type pressure-outlet	ID 10
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh	Periodic Conditions	
Help		

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.

- a. Select pressure-outlet from the Type drop-down list in the Boundary Conditions task page.
- b. Click Yes when ANSYS Fluent asks you if you want to change the zone type.

The **Pressure Outlet** boundary condition dialog box will open.

Pressure Outlet	×
Zone Name	
outlet	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Gauge Pressure (pascal) 0 constant	•
Backflow Direction Specification Method From Neighboring Cell	-
Average Pressure Specification	_
Target Mass Flow Rate	
Turbulence	1
Specification Method Intensity and Hydraulic Diameter	-
Backflow Turbulent Intensity (%) 5	e
Backflow Hydraulic Diameter (mm) 50	P
OK Cancel Help	

- i. Select **From Neighboring Cell** from the **Backflow Direction Specification Method** drop-down list.
- ii. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
- iii. Retain 5% for **Backflow Turbulent Intensity**.
- iv. Enter 50 mm for Backflow Hydraulic Diameter.
- v. Click OK to close the Pressure Outlet dialog box.
- 3. Set the boundary type to **axis** for both the **axis-inlet** and the **axis-move** boundaries.

Organization Boundary Conditions

Since the **axis-inlet** and the **axis-move** boundaries are assigned to a wall boundary type in the original mesh, you will need to explicitly assign these boundaries to an axis boundary type in ANSYS Fluent.

- a. Select axis-inlet from the Zone list and select axis from the Type list.
- b. Click **Yes** when ANSYS Fluent asks you if you want to change the zone type.
- c. Retain the default Zone Name in the Axis dialog box and click OK to close the Axis dialog box.
- d. Select axis-move from the Zone list and select axis from the Type list.
- e. Click Yes when ANSYS Fluent asks you if you want to change the zone type.
- f. Retain the default **Zone Name** in the **Axis** dialog box and click **OK** to close the **Axis** dialog box.

15.4.7. 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

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	<u>.</u>
Least Squares Cell Based 💌	
Pressure	
PRESTO!	Ξ
Density	
Second Order Upwind 🔹	
Momentum	-
Second Order Upwind 🔹	
Turbulent Kinetic Energy	
First Order Upwind 🔹	Ŧ
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation Frozen Transient	
High Order Term Relaxation Options	
Default	
Help	

- 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.

Solution Controls

Solution Controls	
Flow Courant Number	
200	٦
Explicit Relaxation Factors	
Momentum 0.75]
Pressure 0.75	
Under-Relaxation Factors	
Density	4
1	
Body Forces	
1	
Turbulent Kinetic Energy	=
0.8	
Turbulent Dissipation Rate	
0.8	
Turbulent Viscosity	
1	
Default	
Equations Limits Advanced	
Help	

a. Retain the default values for **Under-Relaxation Factors** in the **Solution Controls** task page.

3. Enable the plotting of residuals during the calculation.



Residual Monitors					-X
Options	Equations				
Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria	<u> </u>
V Plot	continuity	V		0.001	
Window	x-velocity			0.001	E
Iterations to Plot	y-velocity			0.001	
1000	energy	V		1e-06	.
	Residual Values			Convergence Cr	riterion
Iterations to Store	Normalize		Iterations	absolute	•
1000			5		
	V Scale				
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 solution.

Over Solution Initialization

Solution Initialization			
Initialization Methods Hybrid Initialization Standard Initialization 			
More Settings Initialize			
Patch			
Reset DPM Sources Reset Statistics			
Help			

- a. Retain the default Hybrid Initialization in the Initialization Methods group box.
- b. Click Initialize in the Solution Initialization task page.

Note

A warning is displayed in the console stating that the convergence tolerance of 1.000000e-06 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 User's Guide.

c. Click More Settings....

U Hybrid Initialization	x
General Settings Turbulence Settings Species Settin	gs I
Number of Iterations 20	
Explicit Under-Relaxation Factor	
Scalar Equation-0	
Scalar Equation-1	
Reference Frame	
 Relative to Cell Zone Absolute 	
Initialization Options	
Use Specified Initial Pressure on Inlets Use External-Aero Favorable Settings Maintain Constant 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 once more.

Note

Click **OK** in the **Question** dialog box, where it asks to discard the current data. The console displays 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).

File \rightarrow Write \rightarrow Case...

6. Start the calculation by requesting 150 iterations.

Calculation

Run Calculation	
Check Case	review Mesh Motion
Number of Iterations Re 150	eporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

Click Calculate.

The solution converges in approximately 115 iterations.

- 7. Save the case and data files (valve_init.cas.gz and valve_init.dat.gz).
 - File \rightarrow Write \rightarrow Case & Data...

15.4.8. Time-Dependent Solution Setup

1. Enable a time-dependent calculation.



General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Type Pressure-Based Density-Based	Velocity Formulation
Time Steady Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

a. Select Transient from the Time list in the General task page.

15.4.9. Mesh Motion

1. Select and compile the user-defined function (UDF).

 $\textbf{Define} \rightarrow \textbf{User-Defined} \rightarrow \textbf{Functions} \rightarrow \textbf{Compiled...}$

Compiled UDFs	— ×-
Source Files	Header Files
Library Name libudf	Build
Load	Help

a. Click Add... in the Source Files group box.

The **Select File** dialog box will open.

- i. Select the source code valve.c in the Select File dialog box, and click OK.
- b. Click **Build** in the **Compiled UDFs** dialog box.

The UDF is already defined, but it must be compiled within ANSYS Fluent before it can be used in the solver. Here you will create a library with the default name of <code>libudf</code> in your working folder. If you want to use a different name, you can enter it in the **Library Name** field. In this case you need to make sure that you will open the correct library in the next step.

A dialog box will appear warning you to make sure that the UDF source files are in the directory that contains your case and data files. Click **OK** in the warning dialog box.

c. Click Load to load the UDF library you just compiled.

When the UDF is built and loaded, it is available to hook to your model. Its name will appear as **valve::libudf** and can be selected from drop-down lists of various dialog boxes.

2. Hook your model to the UDF library.

💶 User-Define	d Function Hooks	-
Initialization	none	Edit
Adjust	none	Edit
Execute at End	none	Edit
Read Case	none	Edit
Write Case	none	Edit
Read Data	reader::libudf	Edit
Write Data	writer::libudf	Edit
Execute at Exit	none	Edit
Wall Heat Flux none -		
OK Cancel Help		

Define \rightarrow **User-Defined** \rightarrow **Function Hooks...**

- a. Click the Edit... button next to Read Data to open the Read Data Functions dialog box.
 - i. Select reader::libudf from the Available Read Data Functions selection list.
 - ii. Click Add to add the selected function to the Selected Read Data Functions selection list.
 - iii. Click OK to close the Read Data Functions dialog box.
- b. Click the Edit... button next to Write Data to open the Write Data Functions dialog box.
 - i. Select writer::libudf from the Available Write Data Functions selection list.
 - ii. Click Add to add the selected function to the Selected Write Data Functions selection list.
 - iii. Click OK to close the Write Data Functions dialog box.

These two functions will read/write the position of the center of gravity (CG) and velocity in the X direction to the data file. The location of the CG and the velocity are necessary for restarting a case. When starting from an intermediate case and data file, ANSYS Fluent needs to know the location of the CG and velocity, which are the initial conditions for the motion calculation. Those values are saved in the data file using the writer UDF and will be read in using the reader UDF when reading the data file.

- c. Click OK to close the User-Defined Function Hooks dialog box.
- 3. Enable dynamic mesh motion and specify the associated parameters.

Dynamic Mesh

Dynamic Mesh		
V Dynamic Mesh		
Mesh Methods	Options	
Smoothing	In-Cylinder	
Layering	Six DOF	
Remeshing	Implicit Update	
Settings	Contact Detection	
	Settings	
Events		
Dynamic Mesh Zones		
Create/Edit	elete Delete All	
Display Zone Motion		
Preview Mesh Motion		
(Tever near modo)		
Help		

a. Enable Dynamic Mesh in the Dynamic Mesh task page.

For more information on the available models for moving and deforming zones, see Modeling Flows Using Sliding and Dynamic Meshes in the User's Guide.

b. Disable Smoothing and enable Layering in the Mesh Methods group box.

ANSYS Fluent will automatically flag the existing mesh zones for use of the different dynamic mesh methods where applicable.

c. Click the **Settings...** button to open the **Mesh Method Settings** dialog box.

💶 Mesh Method Settings 🛛 💽
Smoothing Layering Remeshing
Options Options Height Based Ratio Based Split Factor 0.4
Collapse Factor 0.2
OK Cancel Help

- i. Click the **Layering** tab.
- ii. Select Ratio Based in the Options group box.
- iii. Retain the default settings of 0.4 and 0.2 for Split Factor and Collapse Factor, respectively.
- iv. Click OK to close the Mesh Method Settings dialog box.
- 4. Specify the motion of the fluid region (fluid-move).

Dynamic Mesh → Create/Edit...

The valve motion and the motion of the fluid region are specified by means of the UDF valve.

Dynamic Mesh Zones	×
Zone Names	Dynamic Mesh Zones
fluid-move 🔹	
Туре	
Stationary	
Rigid Body	
Deforming	
 User-Defined System Coupling 	
System Coupling	
Motion Attributes Geometry Definition Meshin	g Options Solver Options
Motion UDF/Profile	
valve::libudf	
Center of Gravity Location	Center of Gravity Orientation
X (mm) 0	Theta_Z (deg)
~ (m) 0	meta_z (deg) 0
Y (mm) 0	
Create Draw Delete All	Delete Close Help

- a. Select **fluid-move** from the **Zone Names** drop-down list.
- b. Retain the default selection of Rigid Body in the Type group box.
- c. Ensure that **valve::libudf** is selected from the **Motion UDF/Profile** drop-down list in the **Motion Attributes** tab to hook the UDF to your model.
- d. Retain the default settings of (0, 0) mm for **Center of Gravity Location**, and 0 for **Center of Gravity Orientation**.

Specifying the CG location and orientation is not necessary in this case, because the valve motion and the initial CG position of the valve are already defined by the UDF.

- e. Click Create.
- 5. Specify the meshing options for the stationary layering interface (**int-layering**) in the **Dynamic Mesh Zones** dialog box.

Dynamic Mesh Zones		—
Zone Names int-layering Type © Stationary © Rigid Body © Deforming © User-Defined © System Coupling	Dynamic Mesh Zones fluid-move	
Motion Attributes Geometry Definition Meshin	g Options Solver Options	
Adjacent Zone fluid-move Co	ell Height (mm) 0.5	constant 👻
Adjacent Zone fluid-inlet Co	ell Height (mm)	constant 🔹
Create Draw	Delete All Delete	Close Help

- a. Select int-layering from the Zone Names drop-down list.
- b. Select Stationary in the Type group box.
- c. Click the **Meshing Options** tab.
 - i. Enter 0.5 mm for Cell Height of the fluid-move Adjacent Zone.
 - ii. Retain the default value of 0 mm for the **Cell Height** of the fluid-inlet **Adjacent zone**.
- d. Click Create.
- 6. Specify the meshing options for the stationary outlet (outlet) in the Dynamic Mesh Zones dialog box.
 - a. Select outlet from the Zone Names drop-down list.
 - b. Retain the previous selection of **Stationary** in the **Type** group box.
 - c. Click the **Meshing Options** tab and enter 1.9 mm for the **Cell Height** of the fluid-move **Adjacent Zone**.
 - d. Click Create.
- 7. Specify the meshing options for the stationary seat valve (**seat-valve**) in the **Dynamic Mesh Zones** dialog box.
 - a. Select **seat-valve** from the **Zone Names** drop-down list.

- b. Retain the previous selection of Stationary in the Type group box.
- c. Click the **Meshing Options** tab and enter 0.5 mm for **Cell Height** of the fluid-move **Adjacent Zone**.
- d. Click Create.
- 8. Specify the motion of the valve (valve) in the Dynamic Mesh Zones dialog box.
 - a. Select **valve** from the **Zone Names** drop-down list.
 - b. Select **Rigid Body** in the **Type** group box.
 - c. Click the **Motion Attributes** tab.
 - i. Ensure that **valve::libudf** is selected from the **Motion UDF/Profile** drop-down list to hook the UDF to your model.
 - ii. Retain the default settings of (0, 0) mm for **Center of Gravity Location**, and 0 for **Center of Gravity Orientation**.
 - 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 MDM problems, you may want to preview the mesh motion before proceeding. In this problem, the mesh motion is driven by the pressure exerted by the fluid on the valve and acting against the inertia of the valve. Hence, for this problem, mesh motion in the absence of a flow field solution is meaningless, and you will not use this feature here.

15.4.10. Time-Dependent Solution

1. Set the solution parameters.

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!	Ξ
Density	
Second Order Upwind 🗸	
Momentum	
Second Order Upwind 🗸	
Turbulent Kinetic Energy	
First Order Upwind 🔹	Ŧ
Transient Formulation	
First Order Implicit 👻	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
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.

Solution Controls

Solution Controls	
Under-Relaxation Factors	
Pressure	_
0.6	
Density	
1	Ξ
Body Forces	
1	
Momentum	-
0.7	
Turbulent Kinetic Energy	
0.4	
	Ŧ
Default	
Equations Limits Advanced	
Help	

- a. Enter 0.6 for Pressure in the Under-Relaxation Factors group box.
- b. Enter 0.4 for Turbulent Kinetic Energy.
- c. Enter 0.4 for **Turbulent Dissipation Rate**.
- 3. Request that case and data files are automatically saved every 50 time steps.

 $\textcircled{Calculation Activities (Autosave Every (Time Steps)) \rightarrow Edit...}$

💶 Autosave 📃
Save Data File Every (Time Steps) 50 🔍
Data File Quantities
Save Associated Case Files
 Only if Modified Each Time
File Storage Options
Retain Only the Most Recent Files Maximum Number of Data Files Only Associated Case Files are Retained
File Name
C:\dynamic_mesh\valve_tran.gz Browse
Append File Name with flow-time
Decimal Places in File Name 6
OK Cancel Help

- a. Enter 50 for Save Data File Every (Time Steps).
- b. Enter valve_tran.gz in the File Name text box.
- 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 animation sequences for the static pressure contour plots and velocity vectors plots for the valve.

\bigcirc Calculation Activities (Solution Animations) \rightarrow Create/Edit...

Use the solution animation feature to save contour plots of temperature every five time steps. After the calculation is complete, you use the solution animation playback feature to view the animated temperature plots over time.

💶 Solu	tion Animation				-X
Animatio	n Sequences 2				
Active	Name	Every	When		*
	pressure	5	Time Step	▼ Defin	e
	vv	5	Time Step	▼ Defin	e
	sequence-3	1	Iteration	▼ Defin	e
	sequence-4		Iteration	▼ Defin	e
	sequence-5		Iteration	▼ Defin	e
OK Cancel Help					

- a. Set Animation Sequences to 2.
- b. Enter pressure in the Name text box for the first animation.
- c. Enter vv in the **Name** text box for the second animation.
- d. Set **Every** to 5 for both animation sequences.

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.

- e. Select **Time Step** from the **When** drop-down list for pressure and vv.
- f. Click the **Define...** button next to pressure to open the **Animation Sequence** dialog box.

Animation Sequence	— ו	
Sequence Parameters Storage Type Name In Memory pressure Metafile Window PPM Image Storage Directory	Display Type Mesh Contours Pathlines Particle Tracks Vectors XY Plot Monitor Monitor Monitor Type Residuals Create Tedit	
OK Cancel Help		

i. Retain the default selection of **Metafile** in the **Storage Type** group box.

Note

If you want to store the plots in a folder other than your working folder, enter the folder path in the **Storage Directory** text box. If this field is left blank (the default), the files will be saved in your working folder (that is, the folder where you started ANSYS Fluent).

- ii. Set **Window** number to 1 and click **Set**.
- iii. Select **Contours** in the **Display Type** group box to open the **Contours** dialog box.

Contours	×
Options	Contours of
V Filled	Pressure
Node Values	Static Pressure
 Global Range Auto Range 	Min Max
Clip to Range	0
Draw Profiles	
Draw Mesh	Surfaces 🔋 🗏 🚍
	axis-inlet
Levels Setup	axis-move
20 (1	default-interior:012
	inlat
Surface Name Pattern	New Surface 🕶
Match	Surface Types
	axis
	clip-surf
	exhaust-fan
	fan 👻
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** dropdown lists.
- C. Click **Display** (Figure 15.3: Contours of Static Pressure at t=0 s (p. 661)).
- D. Close the **Contours** dialog box.

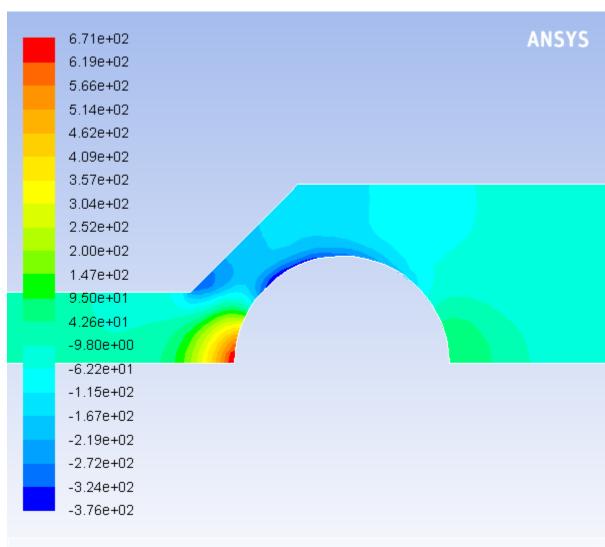


Figure 15.3: Contours of Static Pressure at t=0 s

Contours of Static Pressure (pascal) (Time=0.0000e+00) ANSYS Fluent (axi, pbns, dynamesh, ske, transient)

iv. Click **OK** in the **Animation Sequence** dialog box.

The **Animation Sequence** dialog box will close, and the check box in the **Active** column next to **pressure** in the **Solution Animation** dialog box will be enabled.

- g. Click the **Define...** button next to vv to open the **Animation Sequence** dialog box.
 - i. Retain the default selection of **Metafile** in the **Storage Type** group box.
 - ii. Set Window to 2 and click Set.
 - iii. Select **Vectors** in the **Display Type** group box to open the **Vectors** dialog box.

Vectors		×
Options	Vectors of	
	Velocity	
 Global Range Auto Range 		•
Clip to Range	Color by	
Auto Scale	Velocity	•
Draw Mesh	Velocity Magnitude	•
Style	Min Max	
arrow	0	
Scale Skip		
	Surfaces	
	axis-inlet	<u>^</u>
Vector Options	axis-move default-interior	=
	default-interior:012	
Custom Vectors	inlet	
	int-layering	-
Surface Name Pattern		
Match	New Surface 💌	
	Surface Types	
	axis	*
	clip-surf	
	exhaust-fan	
	fan	-
Display	Compute Close Help	

- A. Retain all the other default settings.
- B. Click **Display** (Figure 15.4: Vectors of Velocity at t=0 s (p. 663)).
- C. Close the **Vectors** dialog box.



Velocity Vectors Colored By Velocity Magnitude (m/s) (Time=0.0000e+00) ANSYS Fluent (axi, pbns, dynamesh, ske, transient)

iv. Click **OK** in the **Animation Sequence** dialog box.

The **Animation Sequence** dialog box will close, and the check box in the **Active** column next to **vv** in the **Solution Animation** dialog box will be enabled.

- h. Click **OK** to close the **Solution Animation** dialog box.
- 5. Set the time step parameters for the calculation.

CRun Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method	Time Step Size (s)
Settings	Number of Time Steps
0-6-6-6	150
Options Extrapolate Variables Data Sampling for Time S Sampling Interval 1 Time Sampled (s	Sampling Options
Max Iterations/Time Step	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- a. Enter 0.0001 s for Time Step Size.
- b. 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.

6. Save the initial case and data files for this transient problem (valve_tran-0.000000.cas.gz and valve_tran-0.000000.dat.gz).

```
\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}
```

7. Request 150 time steps and calculate a solution.

Run Calculation

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.

The UDF (valve.c) that is provided can be edited and customized by changing the parameters as required for your case. In this tutorial, the values necessary for this case were preset in the source code. These values may be modified to best suit your model.

15.4.11. Postprocessing

- 1. Inspect the solution at the final time step.
 - a. Inspect the contours of static pressure in the valve (Figure 15.5: Contours of Static Pressure After 150 Time Steps (p. 666)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

Note

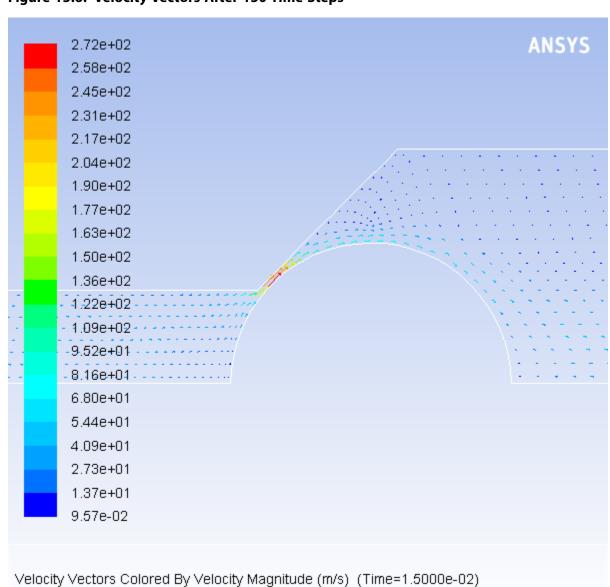
You may need to switch to Window 1 (using the drop-down list at the upper left corner of the graphics window) to view the contour plot.

1.41e+04	ANSYS
1.21e+04	
1.01e+04	
8.07e+03	
6.08e+03	
4.09e+03	
2.10e+03	
1.06e+02	
-1.89e+03	
-3.88e+03	
-5.87e+03	
-7.86e+03	
-9.85e+03	
-1.18e+04	
-1.38e+04	
-1.58e+04	
-1.78e+04	
-1.98e+04	
-2.18e+04	
-2.38e+04	
-2.58e+04	
Contours of Static Pressure (pascal) (Time=1.5000e-02) ANSYS Fluent (axi,	pbns, dynamesh, ske, transient)

Figure 15.5: Contours of Static Pressure After 150 Time Steps

b. Inspect the velocity vectors near the point where the valve meets the seat valve (Figure 15.6: Velocity Vectors After 150 Time Steps (p. 667)).

Graphics and Animations → $\overline{\equiv}$ Vectors → Set Up...



ANSYS Fluent (axi, pbns, dynamesh, ske, transient)

Figure 15.6: Velocity Vectors After 150 Time Steps

2. Play the animation of the pressure contours.

a. \bigcirc Graphics and Animations $\rightarrow \equiv$ Solution Animation Playback \rightarrow Set Up...

Playback	—
Playback	Animation Sequences
Playback Mode Play Once	Sequences pressure
Start Frame Increment End Frame	vv
1 ∢ ► Frame	
	Delete Delete All
	baca baca hi
Write/Record Format Animation Frames	Picture Options
Write Read Close Help	

b. Select pressure from the **Sequences** list in the **Animation Sequences** box of the **Playback** dialog box.

If the **Sequences** list is empty, click **Read...** to select the pressure.cxa sequence file from your working directory.

The playback control buttons will become active.

- c. Set the slider bar above **Replay Speed** about halfway in between **Slow** and **Fast**.
- d. Retain the default settings in the rest of the dialog box and click the 📩 button.

You may have to change the Viewer window to see the animation. In the drop-down menu at the top of the Viewer, set the window number to 1, which corresponds to the **Window** number for pressure that you set in the **Animation Sequence** dialog box.

- 3. Play the animation of the velocity vectors.
 - a. Select vv from the Sequences list in the Animation Sequences box of the Playback dialog box.

If the **Sequences** list does not contain vv, click **Read...** to select the vv. cxa sequence file from your working directory.

b. Retain the default settings in the rest of the dialog box and click the 上 button.

You may have to change the Viewer window to see the animation. In the drop-down menu at the top of the Viewer, set the window number to 2, which corresponds to the **Window** number for vv that you set in the **Animation Sequence** dialog box.

For additional information on animating the solution, see Modeling Transient Compressible Flow (p. 257) and see Animating the Solution of the 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).

 $\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Case \& Data...}$

b. Display the desired contours and vectors.

15.5. 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 (1DoF) rigid body FSI by means of a user-defined function (UDF). ANSYS Fluent can also perform a more general six degrees of freedom (6DoF) rigid body FSI using a built-in 6DoF solver.

If you decide to run this tutorial in parallel, make sure you use **Principal Axes** as the partitioning method.

15.6. 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 16: Modeling Species Transport and Gaseous Combustion

This tutorial is divided into the following sections:

16.1. Introduction
16.2. Prerequisites
16.3. Problem Description
16.4. Background
16.5. Setup and Solution
16.6. Summary
16.7. Further Improvements

16.1. Introduction

This tutorial examines the mixing of chemical species and the combustion of a gaseous fuel.

A cylindrical combustor burning methane ($\rm CH_4$) 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.

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. 123)

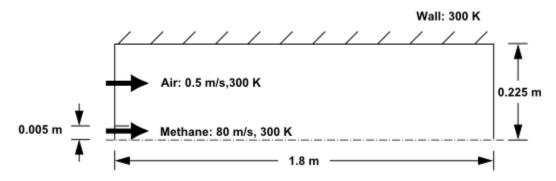
and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

To learn more about chemical reaction modeling, see Modeling Species Transport and Finite-Rate Chemistry in the User's Guide and Species Transport and Finite-Rate Chemistry in the Theory Guide. Otherwise, no previous experience with chemical reaction or combustion modeling is assumed.

16.3. Problem Description

The cylindrical combustor considered in this tutorial is shown in Figure 16.1: Combustion of Methane Gas in a Turbulent Diffusion Flame Furnace (p. 672). 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 $\times 10^3$.





16.4. 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 $CH_4 + 2O_2 \rightarrow CO_2 + 2H_2O$ (16.1)

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.

16.5. Setup and Solution

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

16.5.1. Preparation 16.5.2. Mesh 16.5.3. General Settings 16.5.4. Models 16.5.5. Materials 16.5.6. Boundary Conditions 16.5.7. Initial Reaction Solution 16.5.8. Postprocessing 16.5.9. NOx Prediction

16.5.1. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip species_transport_R150.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 Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Enable **Double-Precision**.
- 11. Ensure Serial is selected under Processing Options.

16.5.2. Mesh

1. Read the mesh file gascomb.msh.

```
\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{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.

16.5.3. General Settings

1. Check the mesh.

$\ \bigcirc \ \ \, \mathsf{General} \to \mathsf{Check}$

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.

Note

ANSYS Fluent will issue a warning concerning the high aspect ratios of some cells and possible impacts on calculation of Cell Wall Distance. The warning message includes recommendations for verifying and correcting the Cell Wall Distance calculation. In this particular case the cell aspect ratio does not cause problems so no further action is required. As an optional activity, you can confirm this yourself after the solution is generated by plotting Cell Wall Distance as noted in the warning message.

2. Scale the mesh.

\$General → Scale...

Since this mesh was created in units of millimeters, you will need to scale the mesh into meters.

💶 Scale N	1esh			—
Domain Ext	ents			Scaling
Xmin (m)	0	Xmax (m)	1.8	Convert Units Service Service
Ymin (m)	0	Ymax (m)	0.225	 Specify Scaling Factors Mesh Was Created In mm
View Lengt	h Unit In			Scaling Factors X 0.001 Y 0.001 Scale Unscale
		C	lose 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 set 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.

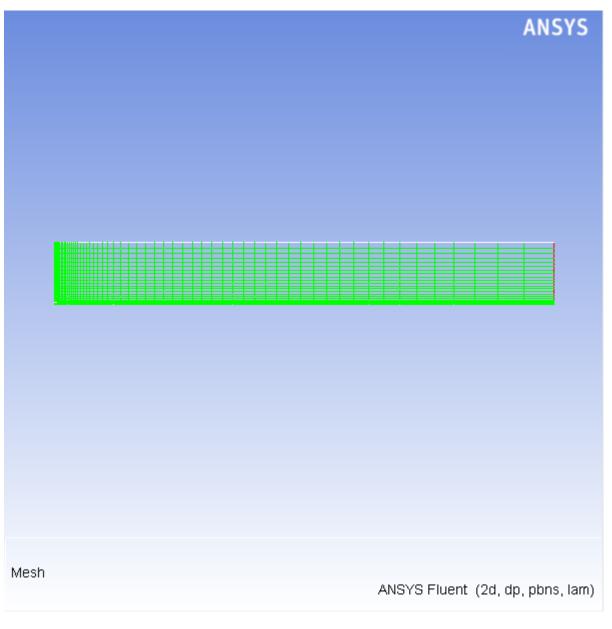
$\clubsuit General \rightarrow Check$

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.





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

General	
Mesh Scale	Check Report Quality
Solver	
Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative
Time ◉ Steady ◯ Transient	2D Space ○ Planar ④ Axisymmetric ○ Axisymmetric Swirl
Gravity	Units
Help	

16.5.4. Models

1. Enable heat transfer by enabling the energy equation.

Models → Energy →	Edit
💶 Energy 💽	
Energy Energy Equation	
OK Cancel Help	

2. Select the standard k- ε turbulence model.



U iscous Model	
Viscous Model Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) k-epsilon Model Standard RNG Realizable Near-Wall Treatment Scalable Wall Functions Scalable Wall Functions Scalable Wall Functions Coptions Viscous Heating Viscous Heating	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 User-Defined Functions User-Defined Functions Trubulent Viscosity none Prandtl Numbers TKE Prandtl Number none TRE Prandtl Number Inone TRR Prandtl Number Inone TDR Prandtl Number Inone Energy Prandtl Number Inone Energy Prandtl Number Inone Energy Prandtl Number Inone Inone Inone
ОК	Cancel Help

a. Select **k-epsilon** 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.

$\textcircled{Models} \rightarrow \overleftarrow{E} Species \rightarrow Edit...$

Species Model	
Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport Reactions Volumetric Wall Surface Particle Surface Options Inlet Diffusion Full Multicomponent Diffusion Thermal Diffusion Relax to Chemical Equilibrium	Mixture Properties Mixture Material methane-air Number of Volumetric Species 5 Turbulence-Chemistry Interaction Laminar Finite-Rate Finite-Rate/Eddy-Dissipation Eddy-Dissipation Eddy-Dissipation Concept Coal Calculator
ОК	Apply Cancel Help

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

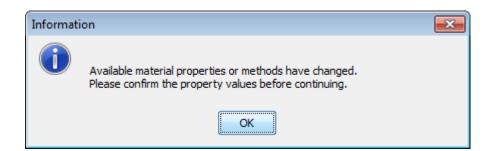
The **Mixture Material** list contains the set of chemical mixtures that exist in the ANSYS Fluent database. You can select one of the predefined mixtures to access a complete description of the reacting system. The chemical species in the system and their physical and thermodynamic properties are defined 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. 680)).

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.

An Information dialog box will open, reminding you to confirm the property values before continuing. Click **OK** *to continue.*



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. 683).

16.5.5. 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.

```
 \diamondsuit Materials \rightarrow \blacksquare Mixture \rightarrow Create/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.

Name		Order Materials by
methane-air	Material Type	 Name
Chemical Formula		Chemical Formula
	FLUENT Mixture Materials	FLUENT Database
	Mixture	User-Defined Database
	none	-
Properties		
Mixture Species	names	
Reaction	eddy-dissipation	
Mechanism	reaction-mechs	
Density (kg/m3)	ncompressible-ideal-gas	
1	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 ch4 o2 co2 h2o n2
Selected Site Species	Add Remove Selected Solid Species
Add Remove	Add Remove
ОК	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.

Reactions	
Mixture methane-air	Total Number of Reactions 1
Reaction Name ID Reaction Type reaction-1 I I Volumetric	🔘 Wall Surface 🛛 🔘 Particle Surface
Number of Reactants 2	Number of Products 2
Species Stoich. Rate Coefficient Exponent Ch4 1 02 2 Arrhenius Rate Pre-Exponential Factor 2.119e+11 Activation Energy (j/kgmol) 2.027e+08 Temperature Exponent 0	Species Stoich. Rate Coefficient Exponent 0 (co2 1 (h2o) 2 0 Mixing Rate A 4 B 0.5
 Include Backward Reaction Third-Body Efficiencies Specify Pressure-Dependent Reaction Specify Coverage-Dependent Reaction Specify 	
ОК	Cancel Help

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 Materials	Chemical Formula
	methane-air	✓ FLUENT Database
	Mixture	User-Defined Database
	none	*
Properties		
Cp (Specific Heat) (j/kg-k) mixing-law	- Edit	
Thermal Conductivity (w/m-k)		
constant	▼Edit	
0.0454		
Viscosity (kg/m-s) constant	Edit	1
1.72e-05	- Curtin	
1.728-05		=
Mass Diffusivity (m2/s) constant-dilute-a	ppx - Edit	
2.88e-05		
		-
Change/Create	Delete Close	Help

- 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.

16.5.6. Boundary Conditions

CBoundary Conditions

Boundary Conditions

Zone		
interior-4		
pressure-outlet-9		
symmetry-5		
velocity-inlet-6		
velocity-inlet-8 wall-2		
wall-7		
WGII-7		
	-	
Phase	Туре	ID
mixture	 symmetry 	5
Edit	Copy Profiles	
Parameters	Operating Conditions	
Farameters	Operating Conditions	
Display Mesh	Periodic Conditions	
\square		
Help		

1. Convert the symmetry zone to the axis type.

↔ Boundary Conditions $\rightarrow \equiv$ 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.

A **Question** dialog box will open, asking if it is OK to change the type of **symmetry-5** from symmetry to axis. Click **Yes** to continue.

Question			×
?	OK to change syn from symmetry to		
	Yes	No	

The **Axis** dialog box will open and display the default name for the newly created axis zone. Click **OK** to continue.

💶 Axis		×
Zone Name		
axis-5		
· · · · · · · · · · · · · · · · · · ·		
	OK Cancel Help	

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	s DPM Multiphase U	DS
Velocity Specification Method	Magnitude, Normal to Boun	ndary 🔻
Reference Frame	Absolute	•
Velocity Magnitude (m/s)	0.5	constant 💌
Supersonic/Initial Gauge Pressure (pascal)	0	constant 💌
Turbulence		
Specification Method I	ntensity and Hydraulic Diam	eter 🔻
	Turbulent Intensity (%	6) 10 P
	Hydraulic Diameter <mark>(</mark> n	n) 0.44 P
OK	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 **Turbulence** 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.

💶 Vel	locity Inlet		×
Zone N	lame		
air-in	let		
Mome	entum Thermal Radiatio	n Species DPM Multiphase UDS	
📃 Sp	pecify Species in Mole Fracti	ons	
	ies Mass Fractions		
ch4	0	constant 👻 🗖	
02	0.23	constant 💌	
co2	0	constant 💌	
h2o	0	constant 💌	
1		*	
		OK Cancel Help	

- h. Click **OK** to close the **Velocity Inlet** dialog box.
- 3. Set the boundary conditions for the fuel inlet (**velocity-inlet-6**).

 $\textcircled{} Boundary \text{ Conditions} \rightarrow \overleftarrow{\sqsubseteq} \text{ velocity-inlet-6} \rightarrow \text{ Edit...}$

Velocity Inlet				
Zone Name				
fuel-inlet				
Momentum Thermal Radiation Species	DPM Multiphase U	DS		
Velocity Specification Method	Magnitude, Normal to Bour	ndary 🔹		
Reference Frame	Absolute	•		
Velocity Magnitude (m/s)	80	constant 🔻		
Supersonic/Initial Gauge Pressure (pascal)	0	constant 💌		
Turbulence				
Specification Method	ntensity and Hydraulic Diam	eter 🔻		
	Turbulent Intensity (%			
	Hydraulic Diameter (n	n) 0.01 P		
OK Cancel Help				

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).

 $\textcircled{P} Boundary \ Conditions \rightarrow \fbox{pressure-outlet-9} \rightarrow \texttt{Edit...}$

Pressure Outlet	×
Zone Name	
pressure-outlet-9	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Gauge Pressure (pascal) 0 constant	•
Backflow Direction Specification Method Normal to Boundary	-
Average Pressure Specification	
Target Mass Flow Rate	
Turbulence	
Specification Method Intensity and Hydraulic Diameter	-
Backflow Turbulent Intensity (%) 10	e
Backflow Hydraulic Diameter (m) 0.45	P
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**).

$\clubsuit Boundary Conditions \rightarrow \blacksquare wall-7 \rightarrow Edit...$

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 Radi	iation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
Heat Flux	Temperature (300	constant 👻
Temperature		Wall Thickness	(m) 0
Convection Radiation			P
Mixed	Heat Generation Rate (w/m3	0	constant 💌
via System Coupling			
Material Name			
aluminum	▼ Edit		
	OK	el Help	
	OK	nep	

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).

$\clubsuit Boundary Conditions \rightarrow \blacksquare wall-2 \rightarrow Edit...$

🛂 Wall	x
Zone Name	
nozzle	
Adjacent Cell Zone	
fluid-1	
Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film	
Thermal Conditions	
	•
Convection Wall Thickness (m)	
CRadiation	
Mixed Heat Generation Rate (w/m3) o constant	-
Material Name	
aluminum 👻 Edit	
OK Cancel Help	

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.

16.5.7. 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

Solution Methods

Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	1
Least Squares Cell Based 🔹	
Pressure	1
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 High Order Term Relaxation Options	
Opdolis	
Set All Species Discretizations Together	
Default	

- 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.

Help

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.

Solution Controls

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	Å.
0.5	
Momentum	Ξ
0.5	
Density	
0.25	
Body Forces	
1	
Turbulent Kinetic Energy	
0.75	-
Default	÷
Equations Limits Advanced	
Set All Species URFs Together	
Help	

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.

tigrid Multi-Stage Exper	t]				
atial Discretization Limiter					
imiter Type					
Standard 👻					
Cell to Face Limiting Cell to Cell Limiting					
eudo Transient Method Usa	ace				
		Inder-Relaxation Factor	Time Scale Factor	_ ^	
Turbulent Kinetic Energy	V	0.8	1	-	
Turbulent Dissipation Rate	V	0.8	1	5	
ch4	V	1	10		
02	V	1	10		
co2	V	1	10	5	
h2o	V	1	10		
Energy	V	1	10	- -	
Default					

- 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.

♦ Monitors → Edit...

Residual Monitors					- X-
Options	Equations				
Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria	A
V Plot	continuity	V	\checkmark	0.001	E
Window 1 Curves 1 Curves Iterations to Plot 1000	x-velocity	V		0.001	
	y-velocity	V		0.001	
	energy			1e-06	Ŧ
	Residual Values			Convergence C	riterion
Iterations to Store	Normalize		Iterations	absolute	•
1000			5		
	Scale				
	Compute Loca	al 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.

Over Solution Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
More Settings Initialize Patch Reset DPM Sources Reset Statistics
Help

- a. Click Initialize to initialize the variables.
- 5. Save the case file (gascombl.cas.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}$

- 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.

CRun Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Pseudo Transient Options	
Fluid Zone	
Time Step Method	Timescale Factor
 User Specified Automatic 	5
Length Scale Method	Verbosity
Aggressive	0
Number of Iterations	Reporting Interval
Data File Quantities	Acoustic Signals
Calculate	
Help	

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.

16.5.8. 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.

Preports → EFIuxes → Set U	lp	
E Flux Reports		
Options Mass Flow Rate Total Heat Transfer Rate Total Sensible Heat Transfer Rate Radiation Heat Transfer Rate Boundary Types axis exhaust-fan fan inlet-vent Boundary Name Pattern Match Save Output Parameter	air-inlet axis-5 fuel-inlet interior-4 nozzle	Results 173.7964051281989 16.64988494053808 0 -11787.00605080781 -192025.3010383084 • •
Compute	ite Close	Help

- a. Select Total Sensible Heat Transfer Rate in the Options list.
- Select all the boundaries from the **Boundaries** selection list (you can click the select-all button (E)). b.
- 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 16.3: Contours of Temperature (p. 697)).

\bigcirc Graphics and Animations → **\sqsubseteq** Contours → Set Up...

- 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**.



	2.31e+03	ANSYS
	2.21e+03	
	2.11e+03	
	2.01e+03	
	1.91e+03	
	1.81e+03	
	1.70e+03	
	1.60e+03	
	1.50e+03	
	1.40e+03	
	1.30e+03	
	1.20e+03	
	1.10e+03	
	9.02e+02	
	8.02e+02	
	7.01e+02	
	6.01e+02 5.01e+02	
	4.00e+02	
	4.00e+02 3.00e+02	
	- 3.000+02	
Cont	ontours of Static Temperature (k)	

ANSYS Fluent (axi, dp, pbns, spe, ske)

The peak temperature is approximately 2310 K.

3. Display velocity vectors (Figure 16.4: Velocity Vectors (p. 699)).

Graphics and Animations → $\stackrel{\frown}{=}$ Vectors → Set Up...

Vectors			— ×
Options	Vectors of		
Global Range	Velocity		•
Auto Range	Color by		
Clip to Range	Velocity		
Auto Scale	velocity		
Draw Mesh	Velocity Magnitude		•
Style	Min	Max	
arrow	0	0	
Scale Skip	Surfaces		
0.01	air-inlet		
	axis-5		
Vector Options	fuel-inlet		E
Custom Vectors	interior-4		
	nozzle		
	outer-wall		-
Surface Name Pattern	New Surface 🔻		
Match	INEW SUFFACE +		
	Surface Types		
	axis		
	clip-surf		
	exhaust-fan		
	fan -		Ψ.
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 Fixed Length X Component Y Component Z Component	Scale Head 0.1 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 16.4: Velocity Vectors

	8.23e+01 A	NSYS
	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	
	3.72e+01	
	3.32e+01	
	2.91e+01	
	2.50e+01	
	2.09e+01	
	1.68e+01	
	1.27e+01	
	8.59e+00	
	4.50e+00	
	4.04e-01	
(also	ait (Venteurs Colleged D. Velenit, Menuitude (m.)	
Velo	city Vectors Colored By Velocity Magnitude (m/s) ANSYS Fluent (axi, dp, pbns, r	spe, ske)
-		-1

4. Display filled contours of stream function (Figure 16.5: Contours of Stream Function (p. 700)).



- a. Select Velocity... and Stream Function from the Contours of drop-down lists.
- b. Click **Display**.

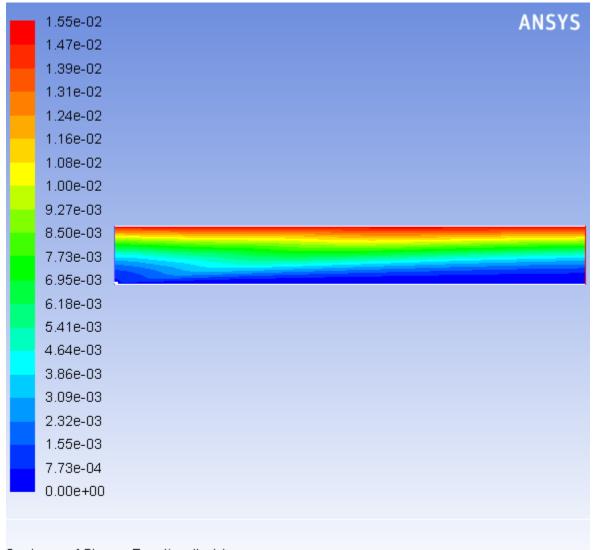


Figure 16.5: Contours of Stream Function

Contours of Stream Function (kg/s)

ANSYS Fluent (axi, dp, pbns, spe, ske)

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_4 (Figure 16.6: Contours of CH4 Mass Fraction (p. 701)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

- a. Select Species... and Mass fraction of ch4 from the Contours of drop-down lists.
- b. Click **Display**.

1.00e+00	ANSYS
9.50e-01	
9.00e-01	
8.50e-01	
8.00e-01	
7.50e-01	
7.00e-01	
6.50e-01	
6.00e-01	
5.50e-01	
5.00e-01	
4.50e-01	
4.00e-01	
3.50e-01	
3.00e-01	
2.50e-01	
2.00e-01	
1.50e-01	
1.00e-01	
5.00e-02	
0.00e+00	
Contours of Moss fraction of ab.4	
Contours of Mass fraction of ch4	ANSYS Fluent (axi, dp, pbns, spe, ske)

Figure 16.6: Contours of CH4 Mass Fraction

6. In a similar manner, display the contours of mass fraction for the remaining species O₂, CO₂, and H₂O (Figure 16.7: Contours of O2 Mass Fraction (p. 702), Figure 16.8: Contours of CO2 Mass Fraction (p. 703), and Figure 16.9: Contours of H2O Mass Fraction (p. 704)) Close the **Contours** dialog box when all of the species have been displayed.

2.30e-01	ANSYS
2.19e-01	
2.07e-01	
1.96e-01	
1.84e-01	
1.73e-01	
1.61e-01	
1.50e-01	
1.38e-01	
1.27e-01	
1.15e-01	
1.04e-01	
9.20e-02	
8.05e-02	
6.90e-02	
5.75e-02	
4.60e-02	
3.45e-02	
2.30e-02	
1.15e-02	
0.00e+00	

Figure 16.7: Contours of O2 Mass Fraction

Contours of Mass fraction of o2

ANSYS Fluent (axi, dp, pbns, spe, ske)

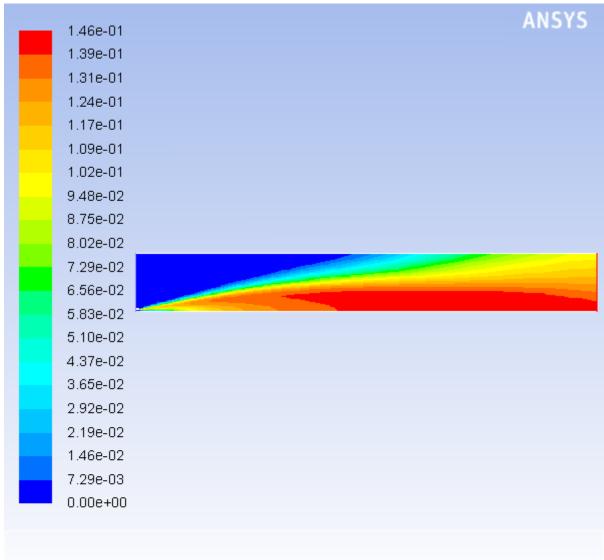


Figure 16.8: Contours of CO2 Mass Fraction

Contours of Mass fraction of co2

ANSYS Fluent (axi, dp, pbns, spe, ske)

4 40- 04	ANSYS
1.19e-01	
1.13e-01	
1.07e-01	
1.01e-01	
9.55e-02	
8.95e-02	
8.36e-02	
7.76e-02	
7.16e-02	
6.57e-02	
5.97e-02	
5.37e-02	
4.78e-02	
4.18e-02	
3.58e-02	
2.98e-02	
2.39e-02	
1.79e-02	
1.19e-02	
5.97e-03	
0.00e+00	

Figure 16.9: Contours of H2O Mass Fraction

Contours of Mass fraction of h2o

ANSYS Fluent (axi, dp, pbns, spe, ske)

7. Determine the average exit temperature.

 $\clubsuit Reports \rightarrow \blacksquare Surface Integrals \rightarrow Set Up...$

Surface Integrals	
Report Type	Field Variable
Mass-Weighted Average 🗸 🗸	Temperature 👻
Surface Types 🔋 🔳 🚍	Static Temperature 🗸 🗸
axis 🔺	Surfaces
exhaust-fan	air-inlet
fan 👻	axis-5
Le .	fuel-inlet interior-4
Surface Name Pattern	nozzle
Match	outer-wall
	pressure-outlet-9
	Mass-Weighted Average (k)
Save Output Parameter	1839.898
	J
Compute Write.	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:

$$\overline{T} = \frac{\int T\rho \, \overline{\nabla} \cdot d \, \overline{A}}{\int \rho \, \overline{\nabla} \cdot d \, \overline{A}} \tag{16.2}$$

- 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.

$$\mathbf{Q}$$
Reports o $oxed{E}$ Surface Integrals o Set Up...

Surface Integrals	
Report Type Area-Weighted Average	Field Variable
Surface Types	Velocity Magnitude
Save Output Parameter	Area-Weighted Average (m/s) 3.30557
Compute Write.	. 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:

$$\overline{v} = \frac{1}{A} \int v dA \tag{16.3}$$

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.

16.5.9. 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.

1. Enable the NOx model.

 $\mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{NOx} \to \mathsf{Edit...}$

NOx Model	
Models Formation Reduction Turbulence Interaction Mode	Formation Model Parameters Thermal Prompt Fuel N2O Path
Pathways Image: Thermal NOx Image: Prompt NOx Image: Fuel NOx Image: Number of Fuel Streams Image: Fuel Stream ID Image: Fuel Stream ID	Fuel Carbon Number 1 Equivalence Ratio 0.76
Apply Close	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		EX
Models Formation Reduction	Turbulence Interaction Mode	Formation Model Parameters Thermal Prompt Fuel N2O Path
PDF Mode PDF Type	temperature beta PDF Points 20	[O] Model partial-equilibrium [OH] Model none
Temperature Variance Tmax Option	transported 👻	I
	Apply Close	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	
	temperature 👻	Fuel Carbon Number	
PDF Type	beta 🔹		
	PDF Points 20	Equivalence Ratio 0.76	
Temperature Variance	transported 🔹		
Tmax Option	global-tmax 👻		
	(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.

\clubsuit Solution Controls \rightarrow Equations...

Equations	-X
Equations	
Flow	
Turbulence	
ch4	
02	
co2	
h2o	
Pollutant no	
Temperature Variance	
Energy	
<u>I</u>	
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**.

✤Solution Controls

a. Set the Time Scale Factor for Pollutant no and Temperature Variance to 10.

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
co2	*
0.75	
h2o	
0.75	
Pollutant no	
1	
Temperature Variance	
1	
Energy	Ξ
0.75	
	Ŧ
Default	
Equations Limits Advanced	
Set All Species URFs Together	
Help	

- i. Click Advanced... to open the Advanced Solution Controls dialog box.
- ii. Enable the pseudo-transient method for **Pollutant no** and **Temperature Variance**, by selecting them under **On/Off** in the **Expert** tab of the **Advanced Solution Controls** dialog box.
- iii. Enter 10 for Time Scale Factor for Pollutant no and Temperature Variance.
- iv. 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.

Monitors →			Edit
------------	--	--	------

Residual Monitors					-X
Options	Equations				
✓ Print to Console	Residual	Monitor (Check Convergence	Absolute Criteria	^
V Plot	pollut_no	V	\checkmark	1e-06	
Window 1 Curves Axes	tvar			0.001	Ŧ
Iterations to Plot	Residual Values			Convergence Cri	iterion
	Normalize		Iterations	absolute	•
These Keese to Oheee	Scale				
Iterations to Store 1000	Compute Loca	al Scale			
OK Plot Renormalize Cancel 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.

CRun Calculation

The solution will converge in approximately 11 iterations.

6. Save the new case and data files (gascomb2.cas.gz and gascomb2.dat.gz).

File \rightarrow Write \rightarrow Case & Data...

7. Review the solution by displaying contours of NO mass fraction (Figure 16.10: Contours of NO Mass Fraction — Prompt and Thermal NOx Formation (p. 713)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

- a. Disable **Filled** in the **Options** group box.
- b. Select NOx... and Mass fraction of Pollutant no from the Contours of drop-down lists.
- c. Click **Display** and close the **Contours** dialog box.

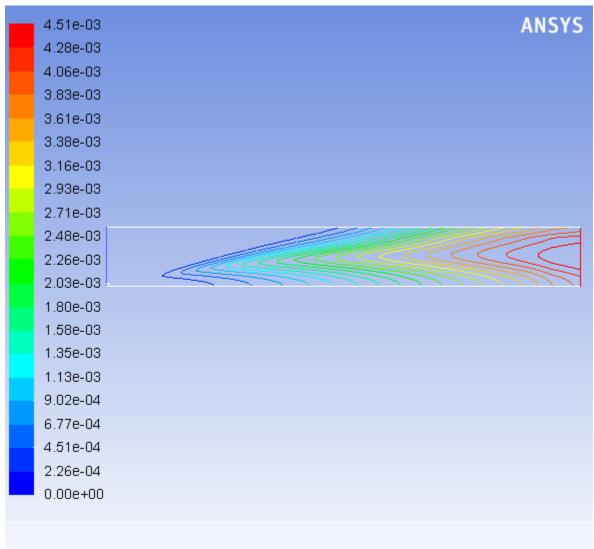


Figure 16.10: Contours of NO Mass Fraction — Prompt and Thermal NOx Formation

Contours of Mass fraction of Pollutant no

ANSYS Fluent (axi, dp, pbns, spe, ske)

8. Calculate the average exit NO mass fraction.

 $\clubsuit Reports \rightarrow \blacksquare Surface Integrals \rightarrow Set Up...$

Surface Integrals	
Report Type Mass-Weighted Average	Field Variable
Surface Types	Mass fraction of Pollutant no
Save Output Parameter	Mass-Weighted Average 0.004213698
Compute Write.	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.

 \bigcirc Models $\rightarrow \equiv$ NOx \rightarrow Edit...

- a. Click the Formation tab and disable Prompt NOx.
- b. Click **Apply** and close the **NOx Model** dialog box.
- 10. Request 25 iterations.

Run Calculation

The solution will converge in approximately 6 iterations.

11. Review the thermal NOx solution by viewing contours of NO mass fraction (Figure 16.11: Contours of NO Mass Fraction—Thermal NOx Formation (p. 715)).

Graphics and Animations → \equiv Contours → Set Up...

- a. Ensure that **NOx...** and **Mass fraction of Pollutant no** are selected from the **Contours of** drop-down list.
- b. Click **Display** and close the **Contours** dialog box.

4.47e-03	ANSYS
4.25e-03	
4.02e-03	
3.80e-03	
3.58e-03	
3.35e-03	
3.13e-03	
2.91e-03	
2.68e-03	
2.46e-03	
2.24e-03	
2.01e-03	
1.79e-03	
1.56e-03	
1.34e-03	
1.12e-03	
8.94e-04	
6.71e-04	
4.47e-04	
2.24e-04	
0.00e+00	
Contours of Mass fracti	ion of Pollutant no
Series of Mass fract	ANSYS Fluent (axi, dp, pbns, spe, ske)

Figure 16.11:	Contours of NO Mass Fraction—Ther	mal 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.

 $\textcircled{Reports} \rightarrow \fbox{Surface Integrals} \rightarrow \texttt{Set Up...}$

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.

 \bigcirc Models $\rightarrow \equiv$ NOx \rightarrow Edit...

- 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.

Run Calculation

The solution will converge in approximately 13 iterations.

15. Review the prompt NOx solution by viewing contours of NO mass fraction (Figure 16.12: Contours of NO Mass Fraction—Prompt NOx Formation (p. 717)).

• Graphics and Animations $\rightarrow \equiv$ Contours \rightarrow Set Up...

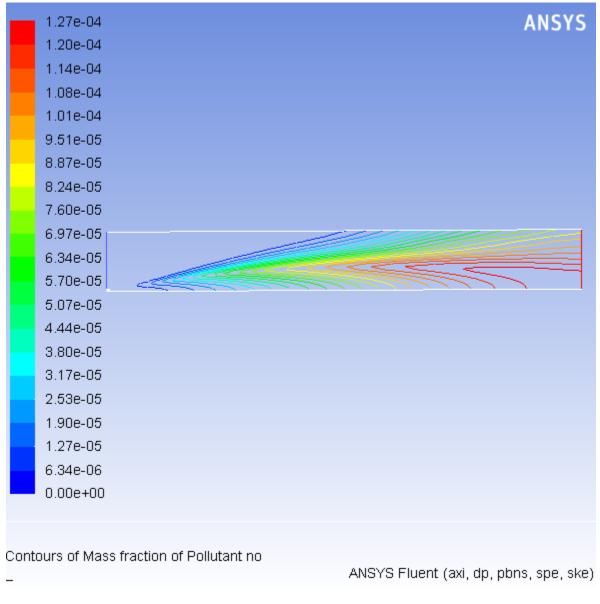


Figure 16.12: Contours of NO Mass Fraction—Prompt NOx Formation

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.

$\textcircled{Reports} \rightarrow \fbox{Surface Integrals} \rightarrow \texttt{Set Up...}$

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:

 $NOppm = \frac{NOmole fraction \times 10^{6}}{1 - H_2 Omole fraction}$

(16.4)

Note

This is the dry ppm. Therefore, the value is normalized by removing the water mole fraction in the denominator.

Define → Custom Field Functions...

💶 Custom Field Function Calculator 🧧	×	
Definition molef-pollut-pollutant-0 * 10 ^ 6 / (1 - molef-h2o) + - X / y^x ABS Select Operand Field Functions from TNN cip cos tap log 10 Field Functions		
INV sin cos tan In log10 Held Functions 0 1 2 3 4 SQRT Species Image: Species Imag		
New Function Name no-ppm		
Define Manage Close Help		

- 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. 123).

- 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 16.13: Contours of NO ppm — Prompt NOx Formation (p. 720)).

Graphics and Animations → \blacksquare Contours → Set Up...

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.

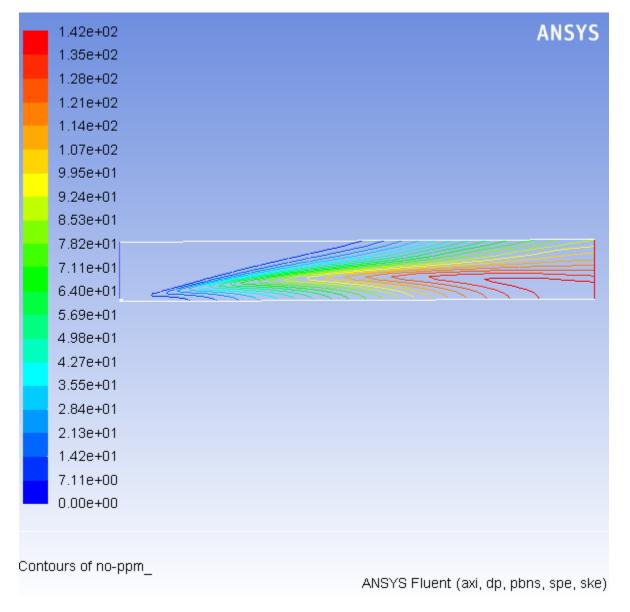


Figure 16.13: Contours of NO ppm — Prompt NOx Formation

The contours closely resemble the mass fraction contours (Figure 16.12: Contours of NO Mass Fraction—Prompt NOx Formation (p. 717)), as expected.

16.6. 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.

16.7. 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 Modeling Non-Premixed Combustion in the 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 σ is the Boltzmann constant (5.729 $\times 10^{-8} W/m^2 - K^4$) 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 Modeling Radiation in the 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. 123).

Chapter 17: Using the Non-Premixed Combustion Model

This tutorial is divided into the following sections:

17.1. Introduction
17.2. Prerequisites
17.3. Problem Description
17.4. Setup and Solution
17.5. Summary
17.6. References
17.7. Further Improvements

17.1. 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- ε 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 β -function for the PDF. For details on the non-premixed combustion modeling approach, see Modeling Non-Premixed Combustion in the User's Guide.

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)

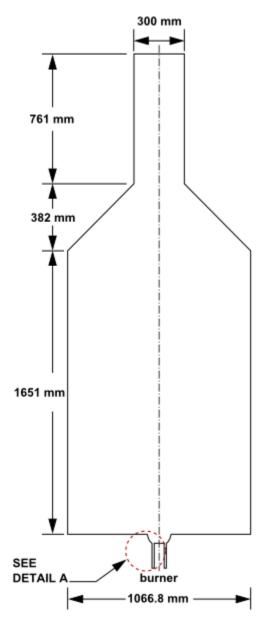
• Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

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

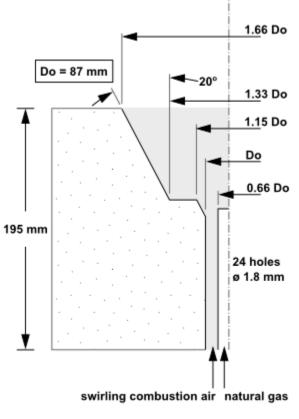
17.3. Problem Description

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 17.1: Problem Description (p. 724), and Figure 17.2: Close-Up of the Burner (p. 725) 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 17.1: Problem Description







DETAIL A

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 Checking the Mesh
- 17.4.3. Specifying Solver and Analysis Type
- 17.4.4. Specifying the Models
- 17.4.5. Defining Materials and Properties
- 17.4.6. Specifying Boundary Conditions
- 17.4.7. Specifying Operating Conditions
- 17.4.8. Obtaining Solution
- 17.4.9. Postprocessing
- 17.4.10. Energy Balances Reporting

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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip non_premix_combustion_R150.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 17.1: Problem Description (p. 724) and Figure 17.2: Close-Up of the Burner (p. 725).

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**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Enable **Double-Precision**.

For more information about Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the Getting Started Guide.

11. Ensure that the Serial processing option is selected.

17.4.2. Reading and Checking the Mesh

1. Read the mesh file berl.msh.

$\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

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. 730).

2. Check the mesh.

$\mathbf{C} General \rightarrow Check$

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.

♀General	\rightarrow	Scale
----------	---------------	-------

Scale Mesh		
Domain Extents		Scaling
Xmin (mm)	Xmax (mm) 2989	Convert Units Specify Scaling Factors
Ymin (mm)	Ymax (mm) 533.4001	Mesh Was Created In
View Length Unit In		Scaling Factors X 0.001 Y 0.001 Scale Unscale
	Close 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.

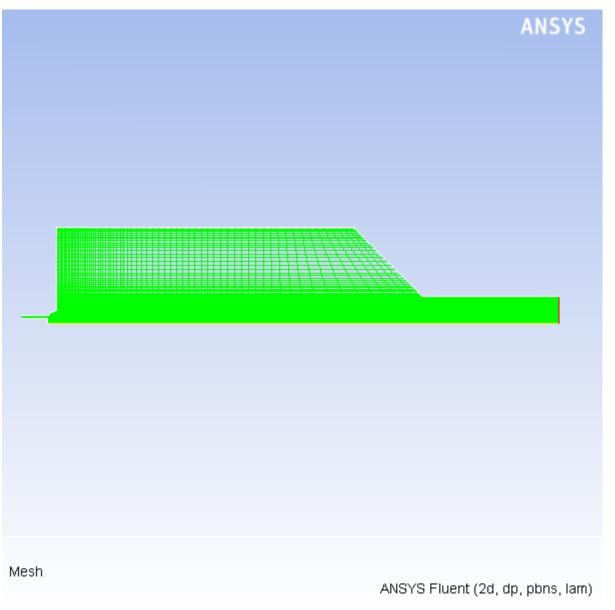
♀General → Check

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 17.3: 2D BERL Combustor Mesh Display (p. 728)).

Figure 17.3: 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.

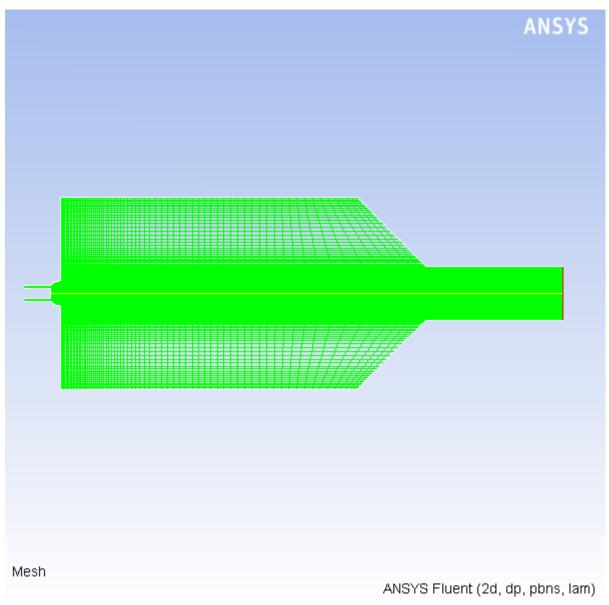
```
Graphics and Animations → Views...
```

D Views		—
Views back front Save Name view-0	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes = axis-2 Define Plane Periodic Repeats Define
Apply Came	ra 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 17.4: 2D BERL Combustor Mesh Display Including the Symmetry Plane (p. 730)





17.4.3. Specifying Solver and Analysis Type

1. Retain the default settings of pressure-based steady-state solver in the **Solver** group box.

⇔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.

General	
Mesh	
Scale Display	Check Report Quality
Solver	
Type Pressure-Based Density-Based	Velocity Formulation
Time ◉ Steady ◯ Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

17.4.4. Specifying the Models

1. Enable the **Energy Equation**.

.

Models → Energy →	Edit
💶 Energy 🛛 💽	
Energy Energy Equation	
OK Cancel Help	

- a. Enable **Energy Equation**.
- b. Click **OK** to close the **Energy** dialog box.

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.

$$\mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Viscous} \to \mathsf{Edit...}$$

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 Treatment 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 Modeling Heat Transfer in the User's Guide.

$\textcircled{Models} \rightarrow \boxed{E} Radiation \rightarrow Edit...$

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) DO/Energy Coupling	Iteration Parameters Energy It Angular Discretization Theta Divisions 2 Phi Divisions 2 Theta Pixels 1 Phi Pixels 1	Non-Gray Model Number of Bands
	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 will list the properties that are required for the model you have enabled. An **Information** dialog box will open, reminding you to confirm the property values.

Informati	on	×
1	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.

 $\textcircled{P} Models \rightarrow \overleftarrow{E} Species \rightarrow Edit...$

Species Model	
Species Model Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport PDF Options Inlet Diffusion Compressibility Effects	PDF Table Creation Chemistry Boundary Control Flamelet Table Properties Premix State Relation Chemical Equilibrium Chemical Equi
	Openaulig Pressure (pascal) 101325 Fuel Stream Rich Flamability Limit 0.064 Thermodynamic Database File Name

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** allows 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					x
Model Off Species Transport Non-Premixed Combustion Premixed Combustion	PDF Table Creation Chemistry Boundary	Control Fla	amelet Table	Properties Premix	1
Premixed Compusition Partially Premixed Combustion Composition PDF Transport PDF Options	02 c2h6	0.017	0.21008		
Inlet Diffusion Compressibility Effects	c3h8	0.001	0	E	
	co2	0.003	0	•	
	Boundary Species Co2 Add Remove List Available Specie	Oxid (k)	315	Specify Species in Mass Fraction Mole Fraction	
	OK Apply	Cancel	Help		

i. Enter c2h6 in the **Boundary Species** text-entry field and click **Add**.

The **c2h6** species will appear 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_2 and 79% $2N_2$ 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

Species	Mole Fraction
n2	0.013
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 Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport PDF Options Inlet Diffusion Compressibility Effects	PDF Table Creation Chemistry Boundary Control Flamelet Table Properties Premix 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 Maximum Number of Species 20 • Minimum Temperature (k) 298 V Automated Grid Refinement Calculate PDF Table Display PDF Table;
	OK Apply Cancel Help

- 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 to open the **PDF Table** dialog box.

PDF Table	×
PDF Data Type Nonadiabatic Table (Two Streams)	
Plot Variable	
Mean Temperature (K)	
Plot Type Options Image: Surface Image: Draw Numbers Box Image: Draw Surface Image: Write To File Surface Parameters	
Constant Value ofSlice byIndexImage: Scaled Heat Loss/Gain Mean Mixture Fraction Scaled VarianceImage: Image: Slice by Image: Image:	
Display Close Help	

- A. Retain the default parameters and click **Display** (Figure 17.5: Non-Adiabatic Temperature Look-Up Table on the Adiabatic Enthalpy Slice (p. 738)).
- B. Close the **PDF Table** dialog box.

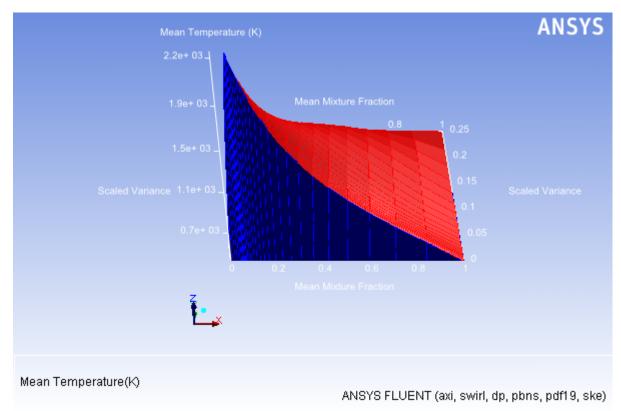


Figure 17.5: Non-Adiabatic Temperature Look-Up Table on the Adiabatic Enthalpy Slice

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 will also be 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.

17.4.5. Defining Materials and Properties

1. Specify the continuous phase (**pdf-mixture**) material.

```
 \diamondsuit Materials \rightarrow \blacksquare pdf-mixture \rightarrow Create/Edit...
```

Name	Material Type		Order Materials by
pdf-mixture	mixture	-	Name Chemical Formula
Chemical Formula	Fluent Mixture Materials		
	pdf-mixture	-	Fluent Database
	Mixture		User-Defined Database
	none	Ŧ	
Properties			
Thermal Conductivity (w/m-k)	constant		
	0.0454		
Viscosity (kg/m-s)	constant		
	1.72e-05		
Absorption Coefficient (1/m)	wsggm-domain-based		
Scattering Coefficient (1/m)	constant		
	0		

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.

a. Select wsggm-domain-based from the Absorption Coefficient drop-down list.

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 of the Theory Guide.

b. Click Change/Create and close the Create/Edit Materials dialog box.

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. The **Density** and **Cp (Specific Heat)** laws cannot be altered: these properties are stored in the non-premixed combustion look-up tables.

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.

17.4.6. 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**).

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{=}$ poutlet-3 \rightarrow Edit...

Pressure Outlet
Zone Name
poutlet-3
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method Normal to Boundary
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) 600
OK Cancel Help

- a. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
- 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).

Velocity Inlet		<u>_</u>
one Name air-inlet-4		
Momentum Thermal Radiation Species	s DPM Multiphase UI	DS
Velocity Specification Method	Components	•
Reference Frame	Absolute	•
Supersonic/Initial Gauge Pressure (pascal)	0	constant 👻
Axial-Velocity (m/s)		vel-prof u 🔻
Radial-Velocity (m/s)	0	constant 💌
Swirl-Velocity (m/s)		vel-prof w
	Swirl Angular Velocity (rad	d/s) 0
Turbulence		
Specification Method I	ntensity and Hydraulic Diam	eter 💌
	Turbulent Intensity (%	6) 17 P
	Hydraulic Diameter (mn	0
OK	Cancel Help	

$\textcircled{Boundary Conditions} \rightarrow \fbox{air-inlet-4} \rightarrow \texttt{Edit...}$

- a. Select Components from the Velocity Specification Method drop-down list.
- b. Select **vel-prof u** from the **Axial-Velocity** drop-down list.
- c. Select **vel-prof w** from the **Swirl-Velocity** drop-down list.
- d. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.
- 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**).

$\textcircled{P} Boundary Conditions \rightarrow \fbox{fuel-inlet-5} \rightarrow \texttt{Edit...}$

Velocity Inlet				
Zone Name				
fuel-inlet-5				
Momentum Thermal Radiation Species	; DPM Multiphase UI	os		
Velocity Specification Method	Components			
Reference Frame	Absolute	•		
Supersonic/Initial Gauge Pressure (pascal)	0	constant 💌		
Axial-Velocity (m/s)	0	constant 💌		
Radial-Velocity (m/s)	157.25	constant 💌		
Swirl-Velocity (m/s) 0 constant				
	Swirl Angular Velocity (rad/s) 0			
Turbulence				
Specification Method [ntensity and Hydraulic Diam	eter 🔻		
	Turbulent Intensity (%	6) 5 P		
	Hydraulic Diameter (mn	n) 1.8		
OK	Cancel Help			

- a. Select Components from the Velocity Specification Method drop-down list.
- b. Enter 157.25 m/s for **Radial-Velocity**.
- c. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list in the **Turbulence** group box.
- 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**.

Q	Boundary	Conditions -	→≣wa	ll-6 → Edit

💶 Wall			×
Zone Name			
wall-6			
Adjacent Cell Zone			
fluid-15			
100 10			
Momentum Thermal Rad	liation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
C Heat Flux	Temperature (k	1370	constant 👻
 Temperature 			
Convection	Internal Emissivit	y 0.5	constant 👻
 Radiation Mixed 		Wall Thickness (m	m)
via System Coupling			P
Material Name	Heat Generation Rate (w/m3	0	constant 🔹
aluminum	▼ Edit		
laidininiani	Lutin		
	OK	Help	

- a. Click the **Thermal** tab.
 - i. Select Temperature in the Thermal Conditions list.
 - 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

Zone Name	Temperature	Internal Emissivity
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**).

Plots →	Frofile	Data →	Set Up
---------	---------	--------	--------

Plot Profile Data		×
Profile vel-prof temp-prof	Y Axis Function	X Axis Function X Y
Plot	Axes Curves Close	Help

- a. Select **temp-prof** from the **Profile** selection list.
- b. Retain the selection of **t** and **x** from the **Y** Axis Function and **X** Axis Function selection lists respectively.
- c. Click Plot (Figure 17.6: Profile Plot of Temperature for wall-9 (p. 745)).

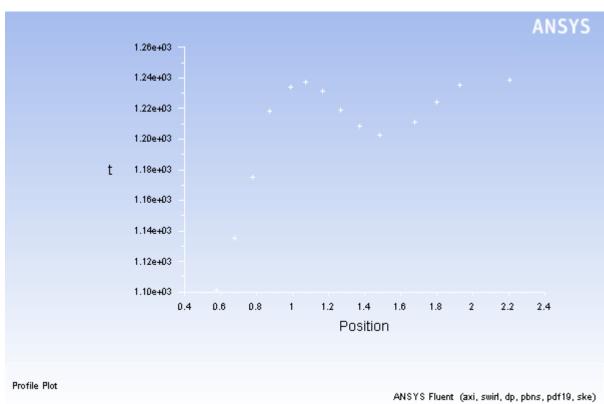


Figure 17.6: Profile Plot of Temperature for wall-9

- 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.

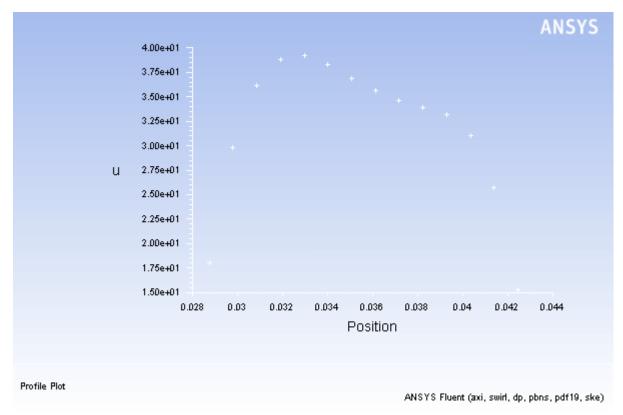
Plot Profile Data		
Profile vel-prof temp-prof	Y Axis Function	X Axis Function
Plot	Axes Curves Close	Help

 $\textcircled{Plots} \rightarrow \overleftarrow{\sqsubseteq} Profile Data \rightarrow Set Up...$

- i. Select **vel-prof** from the **Profile** selection list.
- ii. Retain the selection of **u** from the **Y** Axis Function selection list.
- iii. Select **y** from the **X Axis Function** selection list.

iv. Click Plot (Figure 17.7: Profile Plot of Axial-Velocity for the Swirling Air Inlet (air-inlet-4) (p. 746)).

Figure 17.7: Profile Plot of Axial-Velocity for the Swirling Air Inlet (air-inlet-4)



b. Plot the profile of swirl-velocity for swirling air inlet.

Plot Profile Data		
Profile vel-prof temp-prof	Y Axis Function	X Axis Function x y
Plot	Axes Curves Close	Help

 $\diamondsuit \mathsf{Plots} \to \blacksquare \mathsf{Profile Data} \to \mathsf{Set Up...}$

- i. Retain the selection of **vel-prof** from the **Profile** selection list.
- ii. Select w from the Y Axis Function selection list.
- iii. Retain the selection of **y** from the **X Axis Function** selection list.

iv. Click **Plot** (Figure 17.8: Profile Plot of Swirl-Velocity for the Swirling Air Inlet (air-inlet-4) (p. 747)) and close the **Plot Profile Data** dialog box.

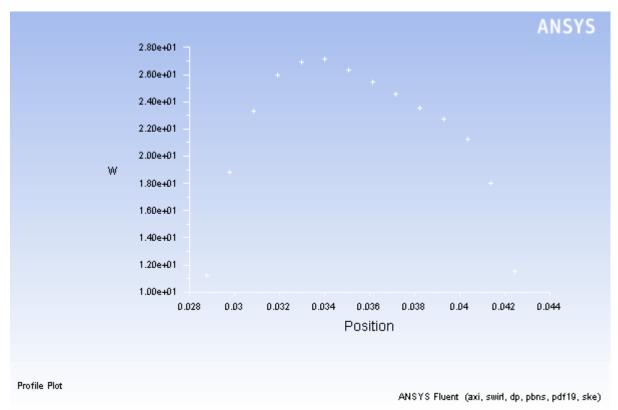


Figure 17.8: Profile Plot of Swirl-Velocity for the Swirling Air Inlet (air-inlet-4)

17.4.7. Specifying Operating Conditions

1. Retain the default operating conditions.

💶 Operati	ng Conditions		×
Pressure		Gravity	
	Operating Pressure (pase 101325	al) 📃 Gravi	ty
Reference	Pressure Location		
X (mm)	0	Đ	
Y (mm)	0	Ð	
Z (mm)	0	μ	
	OK Cancel	Help	

Generating Conditions → Operating Conditions...

The **Operating Pressure** was already set in the PDF table generation in Specifying the Models (p. 731).

17.4.8. Obtaining Solution

1. Set the solution parameters.

✤Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	A
Least Squares Cell Based 🔹	
Pressure	Ξ
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	
Ulinh Order Terr Delevetien	
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.
- c. Retain the other default selections and settings.
- 2. Set the solution controls.

Solution Controls

Solution Controls
Flow Courant Number
70
Explicit Relaxation Factors
Momentum 0.75
Pressure 0.75
Under-Relaxation Factors
Density
0.2
Body Forces
0.8
Swirl Velocity
0.9
Turbulent Kinetic Energy
0.8
Turbulent Dissipation Rate
0.8
Equations Limits Advanced
Help

- 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.

Onitors →	Residuals →	Edit

Residual Monitors					- ×
Options	Equations				
Print to Console	Residual	Monitor	Check Convergence	e Absolute Criteria	A
V Plot	continuity	V		0.001	E
Window	x-velocity			0.001	
Iterations to Plot	y-velocity	V		0.001	-
1000	swirl			0.001	-
	Residual Values			Convergence C	riterion
Iterations to Store	Normalize		Iterations	absolute	•
1000			5		
	Scale				
	Compute Loca	al 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.

Over Solution Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
More Settings Initialize
Reset DPM Sources Reset Statistics

- a. Retain the default selection of Hybrid Initialization from the Initialization Methods group box.
- b. Click **Initialize**.
- 5. Save the case file (berl-1.cas.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}$

6. Start the calculation by requesting 1500 iterations.

CRun Calculation

Run Calculation	
Check Case Preview Mesh Motion	
Number of Iterations Reporting Interval	
Profile Update Interval	
Data File Quantities Acoustic Signals	
Calculate	
Help	

The solution will converge in approximately 830 iterations.

7. Save the converged solution (berl-1.dat.gz).

 $\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Data...}$

17.4.9. Postprocessing

1. Display the predicted temperature field (Figure 17.9: Temperature Contours (p. 753)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

Contours			
Options	Contours of		
V Filled	Temperature 👻		
Vode Values Global Range	Static Temperature		
Auto Range	Min (k) Max (k)		
Clip to Range	309.9997 1979.327		
Draw Mesh	Surfaces 🗦 🗏 🚍		
	air-inlet-4		
Levels Setup	axis-2 default-interior		
20	a fuel-inlet-5		
	poutlet-3		
Surface Name Pattern	New Surface 🔻		
Match	🖳 Surface Types		
	axis		
	dip-surf		
	exhaust-fan		
	fan 👻		
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 1979 K.

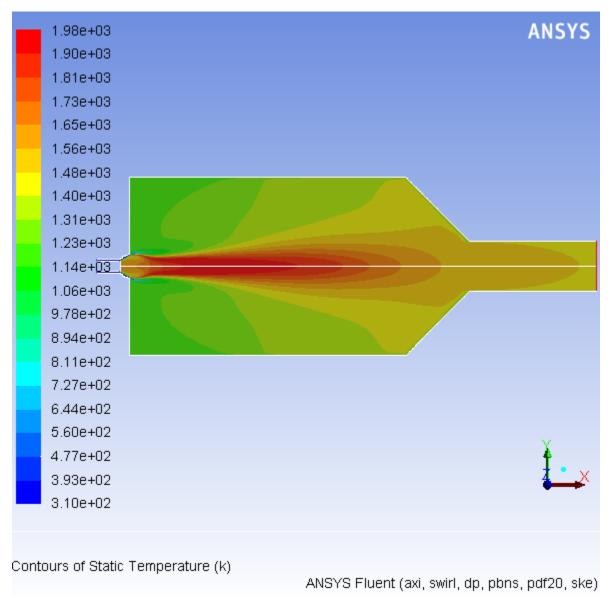


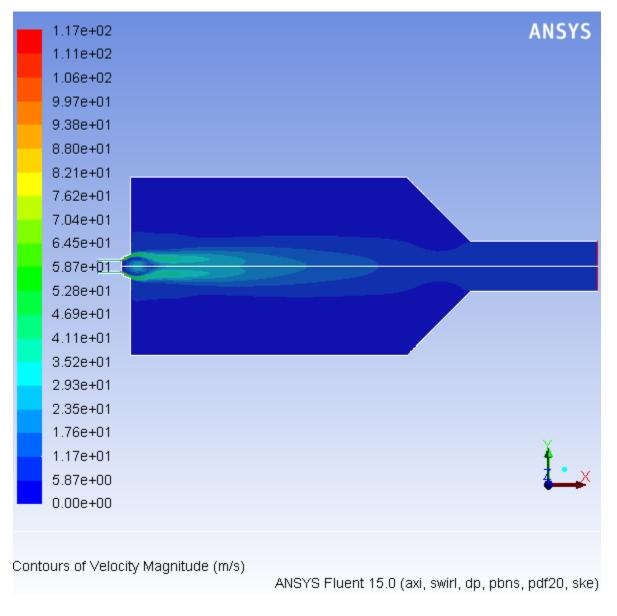
Figure 17.9: Temperature Contours

2. Display contours of velocity (Figure 17.10: Velocity Contours (p. 754)).

\bigcirc Graphics and Animations → **\equiv** Contours → Set Up...

- a. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- b. Click **Display**.

Figure 17.10: Velocity Contours



3. Display the contours of mass fraction of o2 (Figure 17.11: Contours of Mass Fraction of o2 (p. 755)).

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

- a. Select Species... and Mass fraction of o2 from the Contours of drop-down lists.
- b. Click **Display** and close the **Contours** dialog box.

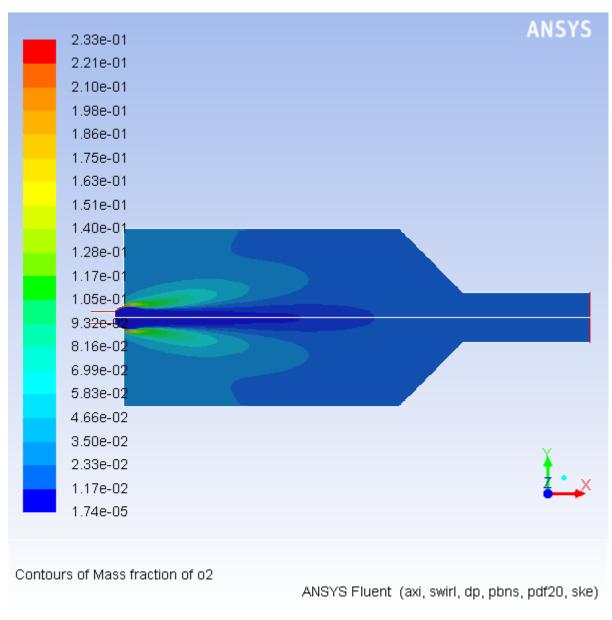


Figure 17.11: Contours of Mass Fraction of o2

17.4.10. 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.

 $\textcircled{P} \mathsf{Reports} \to \fbox{Fluxes} \to \mathsf{Set} \ \mathsf{Up...}$

Flux Reports			—
Options	Boundaries 🗈 🔳		Results
Mass Flow Rate	air-inlet-4	*	0.1194954792235738
Total Heat Transfer Rate	axis-2		
Total Sensible Heat Transfer Rate	default-interior		0.006312729815909114
Radiation Heat Transfer Rate	fuel-inlet-5		-0.1258087639104727
Boundary Types	poutlet-3 wall-10		
axis	wall-11	Ξ	
exhaust-fan	wall-12		
fan	wall-13		
inlet-vent 👻	wall-6		
Boundary Name Pattern	wall-7 wall-8	-	
	wall-9	Ŧ	
Match			
			Net Results (kg/s)
Save Output Parameter			-5.54871e-07
Compute Write	Close	H	elp

- 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.

 $\mathbf{O}_{\mathsf{Reports}} \rightarrow \mathbf{E}_{\mathsf{Fluxes}} \rightarrow \mathsf{Set Up...}$

- 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.

 \clubsuit Reports $\rightarrow \blacksquare$ Surface Integrals \rightarrow Set Up...

Surface Integrals		
Report Type	Field Variable	
Mass-Weighted Average 🗸 🗸	Temperature 👻	
Surface Types	Static Temperature	
axis	Surfaces	
clip-surf exhaust-fan	air-inlet-4	
fan +	axis-2	
	default-interior	
Surface Name Pattern	fuel-inlet-5	
	poutlet-3	
Match	wall-10	
	wall-11	
	wall-12	
	wall-13	
	wall-6	
	wall-7	
	Mass-Weighted Average (k)	
Save Output Parameter	1466.943	
Compute Write 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 1467 K will be displayed in the console.

e. Close the Surface Integrals dialog box.

17.5. 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.

17.6. 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.

17.7. 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. 123).

Chapter 18: Modeling Surface Chemistry

This tutorial is divided into the following sections:

18.1. Introduction18.2. Prerequisites18.3. Problem Description18.4. Setup and Solution18.5. Summary18.6. Further Improvements

18.1. 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 physico-chemical 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.

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. 123)

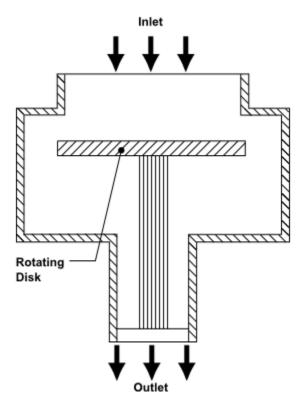
and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

Before beginning with this tutorial, see Modeling Species Transport and Finite-Rate Chemistry in the 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.

18.3. Problem Description

A rotating disk CVD reactor for the growth of Gallium Arsenide (GaAs) shown in Figure 18.1: Schematic of the Reactor Configuration (p. 760) will be modeled.

Figure 18.1: 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.

$AsH_3 + Ga_s \rightarrow Ga + As_s + 1.5H_2$	(18.1)
$Ga(CH_3)_3 + As_s \rightarrow As + Ga_s + 3CH_3$	(18.2)

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 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.

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. Specifying Solver and Analysis Type
- 18.4.4. Specifying the Models
- 18.4.5. Defining Materials and Properties
- 18.4.6. Specifying Boundary Conditions
- 18.4.7. Setting the Operating Conditions
- 18.4.8. Simulating Non-Reacting Flow
- 18.4.9. Simulating Reacting Flow
- 18.4.10. Postprocessing the Solution Results

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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip the surface_chem_R150.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 Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the **Serial** processing option is selected.
- 11. Enable **Double Precision**.

18.4.2. Reading and Checking the Mesh

1. Read in the mesh file surface.msh.

```
\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}
```

2. Check the mesh.

```
\bigcirc General \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.

♀General → Scale...

Scale the mesh to meters as it was created in centimeters.

Scale Mesh		—		
Domain Extents		Scaling		
Xmin (m) -0.2791186	Xmax (m) 0.2791333	 Convert Units Specify Scaling Factors 		
Ymin (m) -0.2793909	Ymax (m) 0.2794	Mesh Was Created In		
Zmin (m)	Zmax (m) 0.454	Scaling Factors		
View Length Unit In	X 0.01			
m 🔻		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.

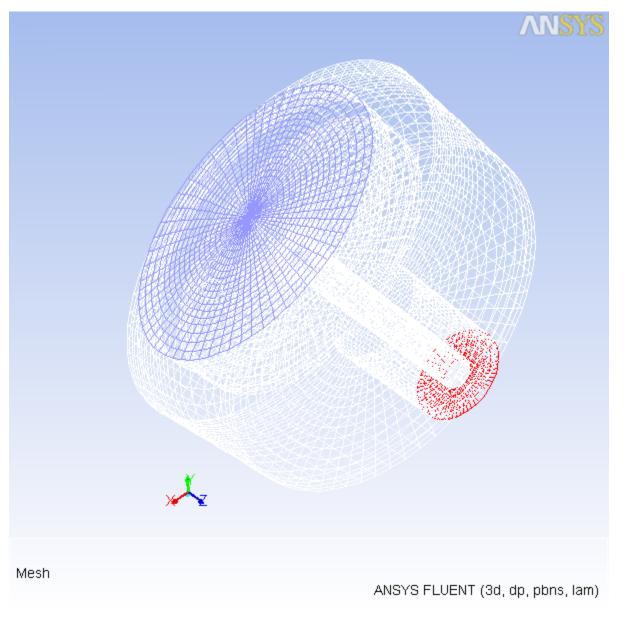
- c. Close the Scale Mesh dialog box.
- 4. Check the mesh.

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 18.2: Mesh Display (p. 764)).

Figure 18.2: 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.

18.4.3. Specifying Solver and Analysis Type

1. Retain the default solver settings of pressure-based steady-state solver in the **Solver** group box.

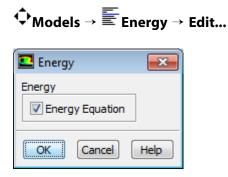
⇔General

General	
Mesh Scale Display	Check Report Quality
Solver	
Type ● Pressure-Based ● Density-Based Time	Velocity Formulation Absolute Relative
 Steady Transient 	
Gravity	Units
Help	

18.4.4. 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 enabling the energy equation.



- a. Enable Energy Equation.
- b. Click **OK** to close the **Energy** dialog box.
- 2. Enable chemical species transport and reaction.

 $\bigcirc Models \rightarrow \blacksquare Species \rightarrow Edit...$

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. 786).

Species Model	
Model	Mixture Properties
 Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport 	Mixture Material mixture-template View Number of Volumetric Species 3 Number of Solid Species 0
Reactions	Number of Site Species
 Volumetric Wall Surface Particle Surface Wall Surface Reaction Options Heat of Surface Reactions Mass Deposition Source Aggressiveness Factor 0 	Turbulence-Chemistry Interaction Caminar Finite-Rate Finite-Rate/Eddy-Dissipation Eddy-Dissipation Eddy-Dissipation Concept Coal Calculator
Options Inlet Diffusion Iffusion Energy Source Ifful Multicomponent Diffusion Iffusion Relax to Chemical Equilibrium Stiff Chemistry Solver CHEMKIN-CFD from Reaction Design	
OK A	pply Cancel Help

a. Select Species Transport in the Model list.

The **Species Model** dialog box will expand to show relevant input options.

- b. Enable Volumetric and Wall Surface in the Reactions group box.
- c. Retain the selection of mixture-template from the Mixture Material drop-down list.

You will modify the mixture material later in this tutorial.

Extra

The **Mixture Material** drop-down list includes all of the chemical mixtures that are currently defined in the ANSYS Fluent database. To check the constituents and the properties of the predefined mixture material, select it from the drop-down list and click the **View...** button next to **Mixture Material** to view a complete description of the reacting system.

- d. Disable Heat of Surface Reactions in the Wall Surface Reaction Options group box.
- e. Enable Mass Deposition Source in the Wall Surface Reaction Options group box.

Mass Deposition Source is enabled because there is a certain loss of mass due to the surface deposition reaction, that is, As(s) and Ga(s) are being deposited out. If you were to do an overall mass balance without taking this fact into account, you would end up with a slight imbalance.

f. 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.

g. 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.

h. Click OK to close the Species Model dialog box.

The ANSYS Fluent console will display a list of the properties that are required for the models that you have enabled.

An *Information* dialog box will open reminding you to confirm the property values that have been extracted from the database.

Informatio	on	×
1	Available material properties or methods have changed. Please confirm the property values before continuing.	
	OK	

i. Click **OK** in the **Information** dialog box.

18.4.5. Defining Materials and Properties

In this step, you will first copy the gas-phase species $(AsH_3, Ga(CH_3)_3, CH_3, and H_2)$ from the ANSYS Fluent database and modify their properties. Then you will 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 Ma-terials** 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), methyl-radical (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 Image: Constraint of the second	Material Type fluid Order Materials by Name Chemical Formula
Properties Cp (Specific Heat) (j/kg-k) piecewise-poly Molecular Weight (kg/kgmol) constant	Nomial View
Standard State Enthalpy (j/kgmol) Standard State Entropy (j/kgmol-k) Constant Constant Constant	▼ View 7 ▼ View
New Edit Save	Copy Close Help

- 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.

f. In the Create/Edit Materials dialog box, select arsenic-trihydride (ash3) from the Fluent Fluid Materials drop-down list.

						×
Vame		Material Type				Order Materials by
arsenic-trihydride		fluid			-	Name
Chemical Formula		- Fluent Fluid Materia	de			Chemical Formula
ash3		arsenic-trihydride			-	Fluent Database
		Mixture	(User-Defined Database
		none			-	
Properties						
Molecular Weight (kg/kgmol)	constant		•]	Edit	^	
	77.95					
Standard State Enthalpy (j/kgmol)	constant		•	Edit		
	0				_	
Standard State Entropy (j/kgmol-k)	constant		•	Edit		
	130579.1		<u>}</u>		E	
Reference Temperature (k)	constant		•	Edit		
	298.15					
I-1 Characteristic Length (angstrom)]	-	

g. In the **Properties** group box, modify the arsenic-trihydride properties as shown in Table 18.1: Properties of Species (p. 769).

Tip

Scroll down in the **Properties** group box to see all the parameters.

Table 18.1: 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
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

Parameter	AsH_3	Ga(CH_3)_3	CH_3	H_2
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

Note

Ignore the **Density** property for now as the density will be set to **incompressible-ideal-gas** for mixture.

h. 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.

i. In a similar way, modify the properties of hydrogen (h2), methyl-radical (ch3), and trimethyl-gallium (game3).

Note

Make sure to click **Change/Create** each time you modify the properties for the material to apply the changes to the local copy.

- 2. Create the site species (Ga_s and As_s) and the solid species (Ga and As).
 - a. 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. Enter the parameter values for the ga_s species as shown in Table 18.2: Properties of Species (p. 770)

Table 18.2: Properties of Species

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 Conductiv- ity	0.0158	0.0158	kinetic-theory	kinetic-theory

Parameter	Ga_s	As_s	Ga	As
Viscosity	2.125e-05	2.125e-05	kinetic-theory	kinetic-theory
Molecular Weight	69.72	74.92	69.72	74.92
Standard State En- thalpy	-3117.71	-3117.71	0	0
Standard State En- tropy	154719.3	154719.3	0	0
Reference Temperat- ure	298.15	298.15	298.15	298.15
L-J Characteristic Length	-	_	0	0
L-J Energy Paramet- er	-	_	0	0

- e. Click **Change/Create** to create the new material.
- f. 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.

g. Create other species 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.
 - $\diamondsuit Materials \rightarrow \blacksquare mixture-template \rightarrow Create/Edit...$
 - 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, Selected Site Species, and Selected Solid 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	
Mixture gaas_deposition	
Available Materials water-vapor (h2o) oxygen (o2) nitrogen (n2) air	Selected Species ash3 game3 ch3 h2
Selected Site Species	Add Remove Selected Solid Species
as_s ga_s	as ga
Add Remove	Add Remove
	Incel

ii. Set the **Selected Species**, **Selected Site Species**, and **Selected Solid Species** from the **Available Materials** selection list as shown in Table 18.3: Selected Species (p. 772)

Selected Species	Selected Site Species	Selected Solid Species
ash3	ga_s	ga
game3	as_s	as
ch3	—	—
h2	—	_

Warning

Ensure that **h2** is at the bottom in the **Selected Species** selection list as shown in Table 18.3: Selected Species (p. 772). ANSYS Fluent will interpret the last species in the list as the bulk species.

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 after all the species are set under the respective categories.
- e. Set the mixture reactions.
 - i. Click the **Edit...** button to the right of the **Reaction** drop-down list to open the **Reactions** dialog box.

Reactions	
Mixture gaas_deposition	Total Number of Reactions 2
Reaction Name ID Reaction Type gallium-dep 1 Image: Comparison of the section Type	Wall Surface O Particle Surface
Number of Reactants 2	Number of Products $\boxed{3}$
Stoich. Rate Coefficient Exponent ash3 1 1 ga_s 1 1 ga_s 1 1 Arrhenius Rate Pre-Exponential Factor 1e+06 Activation Energy (j/kgmol) 0 0 Temperature Exponent 0.5 0.5 Include Backward Reaction Third-Body Efficiencies Specify Pressure-Dependent Reaction Specify Specify Coverage-Dependent Reaction Specify Specify	Species Stoich. Rate ga 1 0 as_s 1 0 wixing Rate 4 8 A 4 8
ОК	Cancel Help

ii. Increase the **Total Number of Reactions** to 2, and define the following reactions using the parameters in Table 18.4: Reaction Parameters (p. 773) : $AsH_3+Ga_s \rightarrow Ga+As_s+1.5H_2$ (18.3)

 $Ga(CH_3)_3 + As_s \rightarrow As + Ga_s + 3CH_3$

Table 18.4: Reaction Parameters

Parameter	For Equation 18.3 (p. 773)	For Equation 18.4 (p. 773)
Reaction Name	gallium-dep	arsenic-dep
Reaction ID	1	2
Reaction Type	Wall Surface	Wall Surface

(18.4)

Parameter	For Equation 18.3 (p. 773)	For Equation 18.4 (p. 773)	
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	PEF= 1e+12, AE= 0, TE= 0.5	
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	as_s= 0, h2 = 0	ga_s= 0, ch3= 0	

Here, PEF = **Pre-Exponential Factor**, AE = **Activation Energy**, and TE = **Temperature Exponent**.

Set the **ID** to 2 in order to set the parameters for the second reaction.

- iii. 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 A Mechanism ID 1 Name gaas-ald						
Reaction Type O Volumetric O Wall Surface All						
Reactions 🖹 🗐 🕅 gallium-dep arsenic-dep	Number of Sites 1	Site Density [kgmol/m2]	Define			
	site-2	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/m² 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	X			
Site Name site-1				
Total Number of Site Species 2				
Site Species Initial Site Coverage	^			
ga_s • 0.7				
as_s • 0.3				
ga_s v				
ga_s v	Ŧ			
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.

~

.

18.4.6. Specifying Boundary Conditions

Boundary Conditions			
Zone			
default-interior outlet			
velocity-inlet			
wall-1			
wall-2 wall-4			
wall-5			
wall-6			
Phase	Туре	ID	
mixture	✓ velocity-inlet	4	
Edit	Copy Profiles		
Parameters	Operating Conditions		
Display Mesh	Periodic Conditions		
Highlight Zone			
Help			

1. Set the conditions for **velocity-inlet**.

 $\clubsuit Boundary \ Conditions \rightarrow \fbox velocity-inlet \rightarrow Edit...$

Velocity Inlet		
Zone Name		
velocity-inlet		
Momentum Thermal Radiation Species	DPM Multiphase U	DS)
Velocity Specification Method	Magnitude, Normal to Boun	idary 👻
Reference Frame	Absolute	•
Velocity Magnitude (m/s)	0.02189	constant 💌
Supersonic/Initial Gauge Pressure (pascal)	0	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 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.

💶 Veloci	ity Inlet	
Zone Nam	e	
velocity	inlet	
		Species DPM Multiphase UDS
	ify Species in Mole Fractions	3
	Mass Fractions	
ash3	0.4	constant 👻 🔶
game3	0.15	constant 👻
ch3	0	constant 👻
1		v
		OK Cancel Help

- f. Click **OK** to close the **Velocity Inlet** dialog box.
- 2. Set the boundary conditions for **outlet**.

$\mathbf{\widehat{\nabla}}$ Boundary Conditions $\rightarrow \mathbf{\overline{\equiv}}$ outlet \rightarrow Edit.

a. Retain the default settings under the **Momentum** tab.

Pressure Outlet	×
Zone Name	
outlet	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Gauge Pressure (pascal) 0 constant	•
Backflow Direction Specification Method Normal to Boundary	-
Radial Equilibrium Pressure Distribution Average Pressure Specification	
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.

💶 Pressu	ure Outlet		
Zone Nam outlet	e		
Moment	um Thermal	Radiation Species DPM Multiphase UDS	1
	ify Species in M Mass Fractions	ole Fractions	
ash3	0.32	constant 🗸	
game3	0.018	constant 🗸	
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			×
Zone Name			
wall-1			
Adjacent Cell Zone			
fluid			
Momentum Thermal R	adiation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions	_		
Heat Flux	Temperature (k	473	constant 💌
Temperature Convection		Wall Thickness ((m)
 Radiation 			P
 Mixed via System Coupling 	Heat Generation Rate (w/m3	0 0	constant 🔹
			Shell Conduction Define
Material Name	▼ Edit		
auminum	← Eat		
	OK Cano	el Help	

- 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**.

♀Boundary Conditions $\rightarrow \stackrel{\frown}{=} wall - 2 \rightarrow Edit...$

- 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.

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{=}$ wall-4 \rightarrow Edit...

💶 Wall					×
Zone Name					
wall-4					
Adjacent Cell Zone					
fluid					
Momentum Thermal	Radiation Species DPM Multiph	ase UDS Wall Film			
Wall Motion	Motion				
 Stationary Wall Moving Wall 	Relative to Adjacent Cell Zone Absolute			Speed (rad/s) 80	P
		Rotation-Axis Origin		Rotation-Axis Direction	
	 Translational Rotational Components 	X (m) 0	P	X	P
	Components	Y (m) 0	P	Y 0	P
		Z (m) 0	Ρ	Z 1	P
Shear Condition					
No Slip Specified Shear Specularity Coeffici Marangoni Stress	ent				
Wall Roughness					
Roughness Height (m)	0 constant	~			
Roughness Constant	0.5 constant	Ţ			
	Ск	Cancel 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	×
Zone Name	
wall-4	
Adjacent Cell Zone	
fluid	
Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film	
Reaction	
Reaction Mechanism gaas-ald	
Surface Area Washcoat Factor 1	
OK Cancel Help	

- 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**.

 $\textcircled{}Boundary \text{ Conditions} \rightarrow \fbox{} wall-5 \rightarrow \texttt{Edit...}$

- 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**.

```
\textcircled{Boundary Conditions} \rightarrow \fbox{wall-6} \rightarrow \texttt{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.

18.4.7. Setting the Operating Conditions

1. Specify the operating conditions.

 $\textcircled{P} Boundary \ Conditions \rightarrow Operating \ Conditions...$

Operating Conditions	
Pressure	Gravity
Operating Pressure (pascal) 10000 P Reference Pressure Location X (m) 0 P Y (m) 0 P Z (m) 0 P	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 C	ancel 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 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 as well as the **Boundary Conditions** task page.

18.4.8. Simulating Non-Reacting Flow

1. Disable Volumetric for solving non-reacting flow.

 \bigcirc Models $\rightarrow \stackrel{\frown}{\equiv}$ Species \rightarrow Edit...

- 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.

CSolution Methods

Solution N	dethods
------------	---------

Scheme	
Coupled	•
Spatial Discretization	
Momentum	-
Second Order Upwind	-
ash3	
Second Order Upwind	
ga <ch3>3</ch3>	
Second Order Upwind	•
dh3	
Second Order Upwind	-
Energy	
Second Order Upwind	• ,
ransient Formulation	
	7
Non-Iterative Time Advancement	_
Frozen Flux Formulation	
Pseudo Transient	
High Order Term Relaxation Options	
Set All Species Discretizations Together	
Default	

- 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.

Controls

Solution Controls	
Flow Courant Number	
200	
Explicit Relaxation Factors	
Momentum 0.75	
Pressure 0.75	
Under-Relaxation Factors	
Density	<u>^</u>
1	
Body Forces	
1	
ash3	Ξ
1	
ga <ch3>3</ch3>	
1	
ch3	
-	-
Default	
Equations Limits Advanced	
Set All Species URFs Together	
Help	

4. Enable residual plotting during the calculation.



- a. Retain the default settings and close the **Residual Monitors** dialog box.
- 5. Initialize the flow field.

Solution Initialization

Solution Initialization
Initialization Methods
 Hybrid Initialization Standard Initialization
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).

```
File \rightarrow Write \rightarrow Case...
```

7. Start the calculation by requesting 200 iterations.

Run Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Number of Iterations	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

a. Enter 200 for Number of Iterations and click Calculate.

The solution will converge in approximately 40 iterations.

18.4.9. Simulating Reacting Flow

1. Enable **Volumetric** for the reacting flow solution.

```
\textcircled{} Models \rightarrow \fbox{} Species \rightarrow Edit...
```

Species Model	
Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport Reactions Volumetric Vall Surface Particle Surface Particle Surface Wall Surface Reaction Options Heat of Surface Reactions Vala Surface Reactions Aggressiveness Factor	Mixture Properties Mixture Material gaas_deposition Image: Species of Control Number of Volumetric Species of Control Number of Solid Species of Control Number of Solid Species of Control Number of Site Species of Control Image: Contret <t< th=""></t<>
Options Inlet Diffusion Diffusion Energy Source Full Multicomponent Diffusion Relax to Chemical Equilibrium Stiff Chemistry Solver CHEMKIN-CFD from Reaction Design OK	pply Cancel Help

- a. Enable Volumetric and Wall Surface in the Reactions group box.
- b. Enable Mass Deposition Source 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.

 $\textcircled{} Monitors \rightarrow \overleftarrow{\blacksquare} Residuals \rightarrow Edit...$

Residual Monitors					- ×
Options	Equations				
Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria	A
V Plot	continuity	V		0.001	
Window 1 Curves Axes	x-velocity			0.001	
Iterations to Plot	y-velocity			0.001	
1000	z-velocity	V		0.001	-
	Residual Values			Convergence C	riterion
Iterations to Store	Normalize		Iterations	absolute	•
1000			5		
	🔽 Scale				
	Compute Loca	al Scale			
OK Plot Renormalize Cancel Help					

3. Request 200 more iterations.

Run Calculation

The solution will converge in approximately 60 additional iterations.

4. Compute the mass fluxes.

✿Reports →	Fluxes → Set Up	
------------	-----------------	--

E Flux Reports		×
Options	Boundaries 🖹 🔳 🚍	Results
 Mass Flow Rate Total Heat Transfer Rate Total Sensible Heat Transfer Rate Radiation Heat Transfer Rate 	default-interior outlet velocity-inlet wall-1	-4.658843801113002e-05 5.596276587044392e-05
Boundary Types	wall-2 wall-4 wall-5 wall-6	-9.37277992099165e-06
Boundary Name Pattern Match Save Output Parameter Compute Write		

- 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 18.3: Contours of Surface Deposition Rate of Ga (p. 790)).

Contours		×
Options	Contours of	
Filled	Species	•
Vode Values Global Range	Surface Deposition Rate of ga	•
Auto Range	Min Max	
Clip to Range	0	
Draw Mesh	Surfaces	
	outlet	
Levels Setup	velocity-inlet	
	wall-1 wall-2	=
	wall-2 wall-4	
	Wall	-
Surface Name Pattern	New Surface	
Ma	Surface Types	
	axis	*
	dip-surf	
	exhaust-fan	
	fan -	Ψ.
Display	Compute Close Help	

Graphics and Animations → \blacksquare Contours → Set Up...

- 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 18.3: Contours of Surface Deposition Rate of Ga (p. 790)).

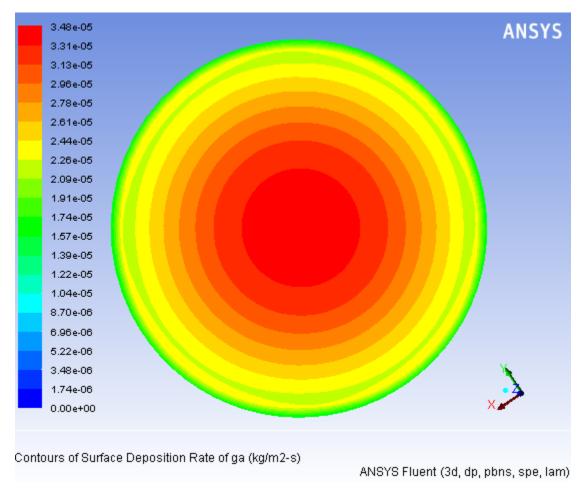


Figure 18.3: Contours of Surface Deposition Rate of Ga

6. Reduce the convergence criteria.

 $\diamondsuit \mathsf{Monitors} \to \mathbf{\overline{\overline{\Xi}}} \mathsf{Residuals} \to \mathsf{Edit...}$

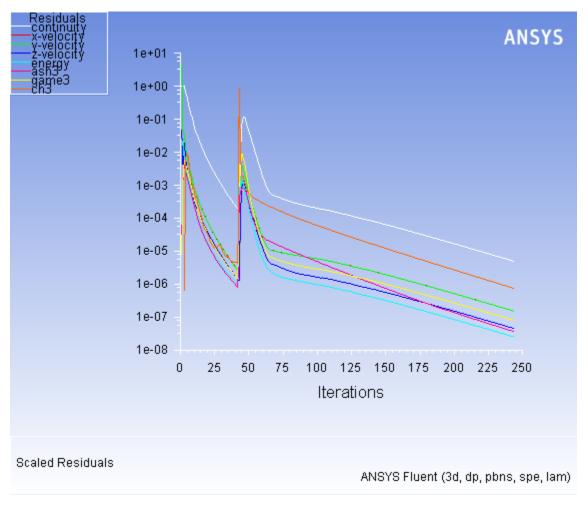
Residual Monitors					×
Options	Equations				_
Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria	*
V Plot	continuity	V		5e-06	
Window 1 Curves Axes	x-velocity	V		0.001	
Iterations to Plot	y-velocity			0.001	
1000	z-velocity	v		0.001	-
	Residual Values			Convergence C	riterion
Iterations to Store	Normalize		Iterations	absolute	•
1000			5		
	Scale				
	Compute Loca	al 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.

CRun Calculation

The solution will converge in approximately 150 additional iterations.

Figure 18.4: Scaled Residuals



8. Check the mass fluxes.



E Flux Reports		— ×-
Options	Boundaries 🖹 🗏	Results
 Mass Flow Rate Total Heat Transfer Rate Total Sensible Heat Transfer Rate Radiation Heat Transfer Rate 	default-interior outlet velocity-inlet wall-1	-4.658467440243972e-05 5.596276587044392e-05
Boundary Types) = wall-2 wall-4	-9.378071349416105e-06
axis exhaust-fan fan inlet-vent	wall-5 wall-6	
Boundary Name Pattern Match		<
Save Output Parameter		Net Results (kg/s) 2.011859e-11
Compute Writ	te Close	Help

- 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...

18.4.10. Postprocessing the Solution Results

1. Create an iso-surface near wall-4.

Surface \rightarrow Iso-Surface...

Iso-Surface		×
Surface of Constant Mesh	From Surface	
Z-Coordinate Image: Constraint of the second s	velocity-inlet wall-1 wall-2 wall-4 wall-5	T T
Iso-Values (m) 0.075438	From Zones 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 18.5: Temperature Contours Near wall-4 (p. 796)).

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

Contours	
Options	Contours of
V Filled	Temperature 👻
Vode Values Global Range	Static Temperature 🗸
Auto Range	Min Max
Clip to Range	0 0
Draw Profiles	Surfaces
	wall-2
	wall-4
Levels Setup	wall-5
20 1 1	wall-6
	₹_0.07
Surface Name Pattern	New Surface -
Match	Surface Types 🗵 🗏 🗏
	axis
	clip-surf
	exhaust-fan fan v
	d
Display	Compute 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**.

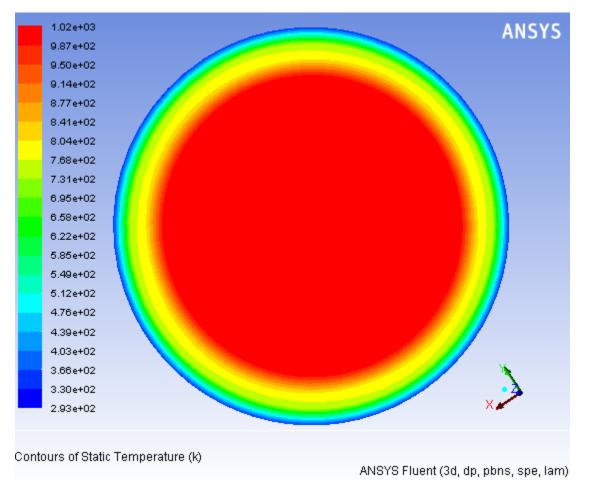


Figure 18.5: Temperature Contours Near wall-4

Figure 18.5: Temperature Contours Near wall-4 (p. 796) 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 18.6: Contours of Surface Deposition Rate of ga (p. 797)).

• Graphics and Animations $\rightarrow \equiv$ Contours \rightarrow Set Up...

- 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 18.6: Contours of Surface Deposition Rate of ga (p. 797) shows the gradient of surface deposition rate of ga. The maximum deposition is seen at the center of the disk.

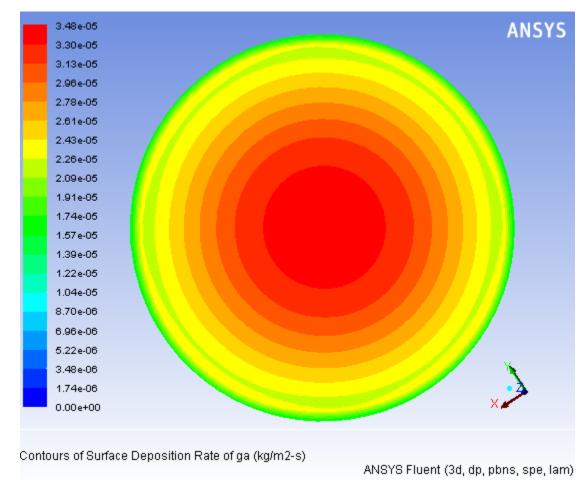


Figure 18.6: Contours of Surface Deposition Rate of ga

4. Display contours of surface coverage of ga_s (Figure 18.7: Contours of Surface Coverage of ga_s (p. 798)).



- 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.

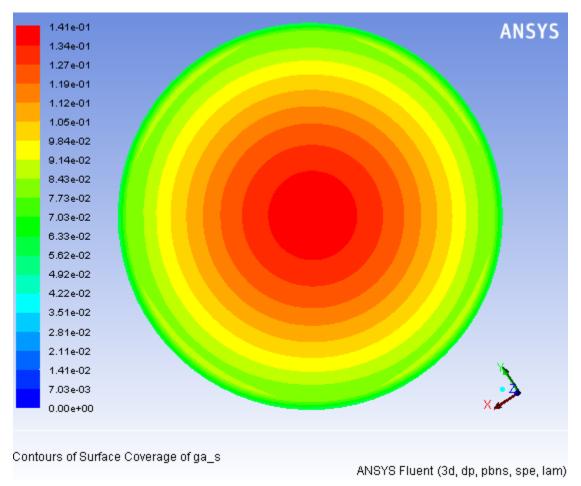




Figure 18.7: Contours of Surface Coverage of $ga_s(p. 798)$ shows the rate of surface coverage of the site species **ga_s**.

5. Create a line surface from the center of **wall-4** to the edge.

Surface → Line/Rake...

Line/Rake Surface	
Options Type Line Tool Reset	Number of Points
End Points	
x0 (m) -0.01040954	x1 (m) 0.1428
y0 (m) -0.004949478	y1 (m) 0.1386585
z0 (m) 0.07620001	z1 (m) 0.07620001
Select Points	with Mouse
New Surface Name	
line-9	
Create Manage	Close Help

a. Enter the values for x0, x1, y0, y1, z0, and z1 as shown in the Line/Rake Surface dialog box.

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. Accept the default name of line-9 for the **New Surface Name** and click **Create**.

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 18.8: Plot of Surface Deposition Rate of Ga (p. 801)).

 $\textcircled{Plots} \rightarrow \overleftarrow{\blacksquare} XY Plot \rightarrow Set Up...$

Solution XY Plot		×
Options Node Values Position on X Axis Position on Y Axis Write to File Order Points	Plot Direction X 1 Y 0 Z 0 Load File Free Data	Y Axis Function Species Surface Deposition Rate of ga X Axis Function Direction Vector Surfaces Cefault-interior Ine-9 outlet velocity-inlet wall-1 wall-2 wall-4 wall-5 New Surface ▼
Plot	Axes	Curves Close Help

- 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*).

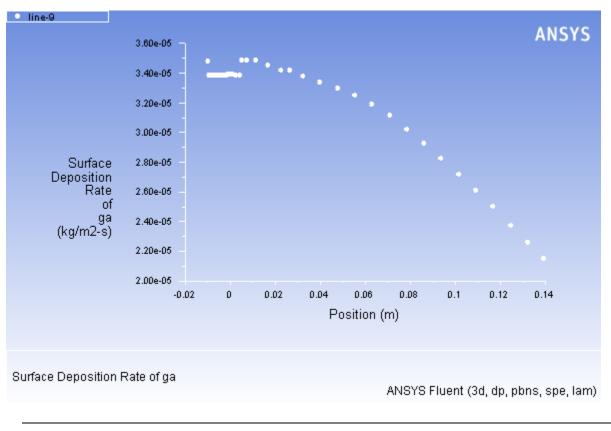


Figure 18.8: Plot of Surface Deposition Rate of Ga

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...
```

18.5. 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.

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. 123).

Chapter 19: Modeling Evaporating Liquid Spray

This tutorial is divided into the following sections:

19.1. Introduction
19.2. Prerequisites
19.3. Problem Description
19.4. Setup and Solution
19.5. Summary
19.6. Further Improvements

19.1. 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.

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:

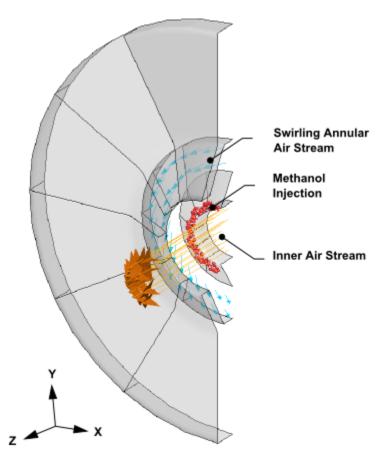
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

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

19.3. Problem Description

The geometry to be considered in this tutorial is shown in Figure 19.1: Problem Specification (p. 804). Methanol is cooled to -10° 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 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. Reading the Mesh
19.4.3. General Settings
19.4.4. Specifying the Models
19.4.5. Materials
19.4.6. Boundary Conditions
19.4.7. Initial Solution Without Droplets
19.4.8. Create a Spray Injection
19.4.9. Solution
19.4.10. Postprocessing

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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip evaporate_liquid_R150.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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Ensure that the **Serial** processing option is selected.
- 11. Ensure that **Double Precision** is disabled.

19.4.2. Reading the Mesh

1. Read in the mesh file sector.msh.

```
\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}
```

2. Change the periodic type of **periodic-a** to rotational.

 $\textcircled{Boundary Conditions} \rightarrow \fbox{periodic-a} \rightarrow \texttt{Edit...}$

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.

19.4.3. General Settings

1. Check the mesh.

\bigcirc General \rightarrow Check

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.

2. Display the mesh.



💶 Mesh Displa	у		×
Options	Edge Type	Surfaces	
Nodes		central_air	•
V Edges	Feature	co-flow-air	
Faces	Outline	default-interior	
Partitions		outer-wall	=
Shrink Factor	Feature Angle	outlet periodic-a	
0	20	periodic-b	
		in	Ψ.
Surface Name Pa	attern Match	New Surface 🔻	
		Surface Types	
Outline Inter	rior	axis	
		clip-surf	
		exhaust-fan	
		fan	*
	Display Colo	rs Close Help	

a. Enable Faces 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 unshaded button (=) at the top of the **Surfaces** selection list, and then select the desired surfaces from the **Surfaces** selection list.

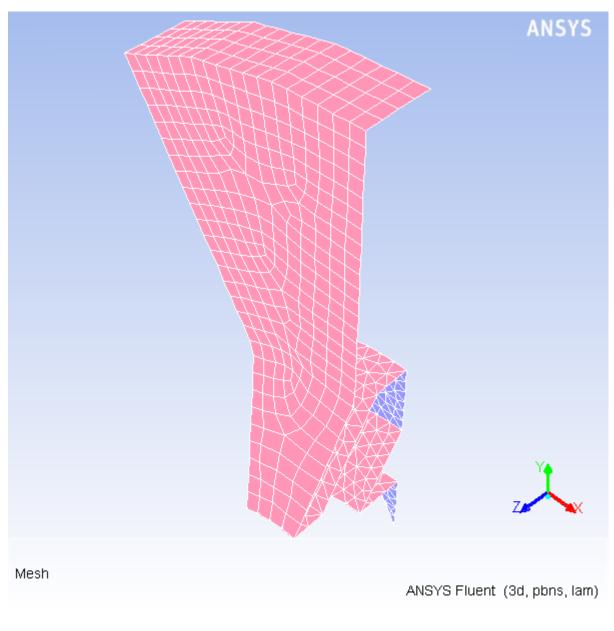
c. Click the Colors... button to open the Mesh Colors dialog box.

Mesh Colors			×
Options © Color by Type © Color by ID Sample	Types far-field inlet interior outlet periodic rans-les-interface symmetry axis wall free-surface internal traction	Colors light gray light green light red light yellow magenta maroon orange pink red tan white yellow	4 III >
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 19.2: Air-Blast Atomizer Mesh Display (p. 808).

Figure 19.2: Air-Blast Atomizer Mesh Display



3. Reorder the mesh using the **Mesh** menu that is found at the top of the ANSYS Fluent window.

$Mesh \rightarrow Reorder \rightarrow Domain$

To speed up the solution procedure, the mesh should be reordered, which will substantially reduce the bandwidth.

ANSYS Fluent will report the progress in the console:

```
Reordering domain using Reverse Cuthill-McKee method:
    zones, cells, faces, done.
    Bandwidth reduction = 30882/741 = 41.68
Done.
```

4. Retain the default solver settings of pressure-based steady-state solver in the **Solver** group box.

General

General
Mesh Scale Check Report Quality Display
Solver
Type Velocity Formulation Image: Pressure-Based Image: Pressure-Based Image: Density-Based Image: Pressure-Based
Time ◉ Steady ◯ Transient
Gravity Units
Help

19.4.4. Specifying the Models

1. Enable heat transfer by enabling the energy equation.

Cancel

OK

2. Enable the Realizable k- ε turbulence model.

Help



E Viscous Model	
	Model Constants C2-Epsilon 1.9 TKE Prandtl Number 1 TDR Prandtl Number
 Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) Large Eddy Simulation (LES) k-epsilon Model Standard RNG Realizable 	1.2 Energy Prandtl Number 0.85 User-Defined Functions Turbulent Viscosity none
Near-Wall Treatment Standard Wall Functions Scalable Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment User-Defined Wall Functions Options Viscous Heating Curvature Correction Production Limiter	Prandtl Number TKE Prandtl Number none TDR Prandtl Number none Energy Prandtl Number none
ОК	Cancel Help

- a. Select k-epsilon (2 eqn) in the Model list.
- b. Select Realizable in the k-epsilon Model list.

The Realizable k- ε model gives a more accurate prediction of the spreading rate of both planar and round jets than the standard k- ε 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.

 $\mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Species} \to \mathsf{Edit...}$

Species Model	
Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport Reactions Volumetric Options Indet Diffusion	Mixture Properties Mixture Material methyl-alcohol-air Number of Volumetric Species 5 5
 Inlet Diffusion Diffusion Energy Source Full Multicomponent Diffusion Thermal Diffusion 	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.

In the console window, ANSYS Fluent lists the properties that are required for the models you have enabled. An **Information** dialog box opens, reminding you to confirm the property values that have been extracted from the database.



d. Click **OK** in the **Information** dialog box to continue.

19.4.5. 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.



ame	Material Type		Order Materials by
methyl-alcohol-air	mixture	•	Name
hemical Formula	Fluent Mixture Materials		Chemical Formula
	methyl-alcohol-air	-	Fluent Database
	Mixture		User-Defined Database
	none	-	
operties			
Mixture Species	names) î	
Density (kg/m3)	ncompressible-ideal-gas	E	
Cp (Specific Heat) (j/kg-k)	nixing-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 air carbon-dioxide (co2) water-vapor (h2o)	Selected Species ch3oh o2 n2
Selected Site Species	Add Remove Selected Solid Species
Add Remove	Add Remove
ОК Са	ncel 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.

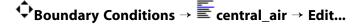
Note

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.

19.4.6. Boundary Conditions

Boundary Conditions	
Zone	
atomizer-wall central_air co-flow-air default-interior outer-wall outlet periodic-a periodic-b swirling_air	
Phase Type ID ID II 11	
Edit Copy Profiles Parameters Operating Conditions Display Mesh Periodic Conditions Highlight Zone	
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 Multiobase U	ns l		
Reference Frame				
Mass Flow Specification Method	Mass Flow Rate	•		
Mass Flow Rate (kg/s)	9.167e-5	constant 🔻		
Supersonic/Initial Gauge Pressure (pascal)	0	constant 🔻		
Direction Specification Method	Direction Vector	•		
Coordinate System	Coordinate System Cartesian (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 I	intensity and Hydraulic Diam	eter 🔹		
	Turbulent Intensity (%	6) 10 P		
	Hydraulic Diameter (n	n) 0.0037 P		
	Cancel Help			

- 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).

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{\equiv}$ co-flow-air \rightarrow Edit...

Velocity Inlet
Zone Name
co-flow-air
Momentum Thermal Radiation Species DPM Multiphase UDS
Velocity Specification Method Magnitude, Normal to Boundary
Reference Frame Absolute
Velocity Magnitude (m/s) 1 constant
Supersonic/Initial Gauge Pressure (pascal) 0 constant
Turbulence
Specification Method Intensity and Hydraulic Diameter
Turbulent Intensity (%) 5
Hydraulic Diameter (m) 0.0726
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).

Pressure Outlet
Zone Name
outlet
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 0 constant
Backflow Direction Specification Method From Neighboring Cell
Radial Equilibrium Pressure Distribution
Average Pressure Specification
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Viscosity Ratio
Backflow Turbulent Intensity (%) 5
Backflow Turbulent Viscosity Ratio 5
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).

$\textcircled{Boundary Conditions} \rightarrow \fbox{swirling_air} \rightarrow \texttt{Edit...}$

Velocity Inlet		—		
Zone Name				
swirling_air				
Momentum Thermal Radiation Species DPM Multiphase UDS				
Velocity Specification Method Magnitude and Direction				
Reference Frame	Absolute	•		
Velocity Magnitude (m/s)	19	constant 🔹		
Supersonic/Initial Gauge Pressure (pascal)	0	constant 💌		
Coordinate System	Cylindrical (Radial, Tangen	tial, Axial) 🔹		
Radial-Component of Flow Direction	0	constant 💌		
Tangential-Component of Flow Direction	0.7071	constant 💌		
Axial-Component of Flow Direction	0.7071	constant 💌		
Turbulence				
Specification Method I	ntensity and Hydraulic Diam	eter 👻		
	Turbulent Intensity (%			
	Hydraulic Diameter (n	n) 0.0043 P		
ОК	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**).

 $\textcircled{P} Boundary Conditions \rightarrow \fbox{outer-wall} \rightarrow \texttt{Edit...}$

🖸 Wali			—
Zone Name			
outer-wall			
Adjacent Cell Zone			
fluid			
Momentum Thermal Rad	iation Species DPM Multiphas	e UDS Wall Film	
Wall Motion Mot	on		
Stationary Wall	Relative to Adjacent Cell Zone		
Moving Wall			
Shear Condition	Shear Stress		
No Slip	X-Component (pascal)	constant	al l
 Specified Shear Specularity Coefficient 			
 Marangoni Stress 	Y-Component (pascal)	constant	
	Z-Component (pascal)	constant	al l
Wall Roughness			
Roughness Height (m) 0	constant	•	
Roughness Constant 0.			
0.	5 constant	•	
	ОКС	ancel 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.

19.4.7. Initial Solution Without Droplets

The airflow will first be solved and analyzed without droplets.

1. Set the solution method.

Solution Methods

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	
v	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
V Pseudo Transient	
High Order Term Relaxation Options	
Set All Species Discretizations Together	
Default	

- a. Select **Coupled** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
- b. Retain the default Second Order Upwind for Momentum.
- c. Enable Pseudo Transient.

Help

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

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	<u>.</u>
0.5	
Momentum	Ξ
0.5	
Density	
1	
Body Forces	
1	
Turbulent Kinetic Energy	
0.75	-
Default Equations Limits Advanced Set All Species URFs Together	
Help	

3. Enable residual plotting during the calculation.

♦ Monitors → Residuals → Edit...

Residual Monitors					x
Options	Equations				
Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria	<u> </u>
V Plot	continuity	v		0.001	E
Window 1 ▲ Curves Axes	x-velocity			0.001]
Iterations to Plot	y-velocity	V		0.001	
1000	z-velocity			0.001	Ŧ
	Residual Values			Convergence Cr	iterion
Iterations to Store	Normalize		Iterations 5	absolute	•
	Scale	al Scale			
OK Plot	Renormaliz	e (Cancel He	lp	

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.

Over Solution Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
More Settings Initialize
Patch
Reset DPM Sources Reset Statistics
Help

- a. Retain the default Hybrid Initialization from the Initialization Methods group box.
- 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).

File \rightarrow Write \rightarrow Case...

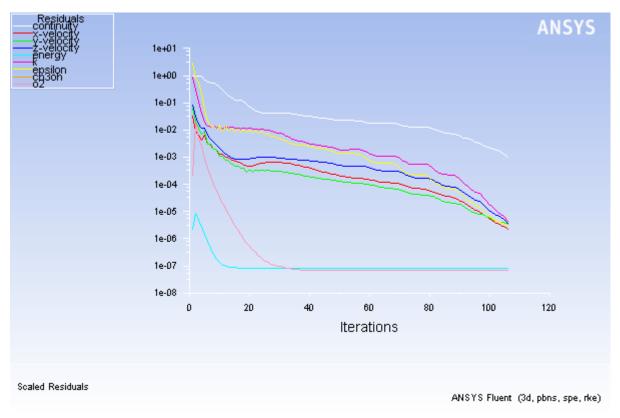
6. Start the calculation by requesting 150 iterations.

Run Calculation

- 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 110 iterations.

Figure 19.3: Scaled Residuals



7. Save the case and data files (spray1.cas.gz and spray1.dat.gz).

```
File \rightarrow Write \rightarrow Case & Data...
```

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.

Surface \rightarrow Iso-Surface...

Iso-Surface		×
Surface of Constant Mesh Angular Coordinate Min (deg) Max (deg) 0 30.00001 Iso-Values (deg) 15 <	From Surface atomizer-wall central_air co-flow-air default-interior outer-wall outlet From Zones 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 19.4: Velocity Magnitude at Mid-Point of Atomizer Section (p. 826)).

Graphics and Animations → \blacksquare Contours → Set Up...

Contours	×
Options	Contours of
V Filled	Velocity 👻
Vode Values Global Range	Velocity Magnitude
Auto Range	Min Max
Clip to Range	0 0
Draw Profiles V Draw Mesh	Surfaces
V Draw Mesi	angle=15
	atomizer-wall
Levels Setup	central air
20 🔺 1	■ co-flow-air
	default-interior 👻
Surface Name Pattern	New Surface 🕶
Mate	h Surface Types
	axis
	clip-surf
	exhaust-fan fan
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 19.4: Velocity Magnitude at Mid-Point of Atomizer Section (p. 826).

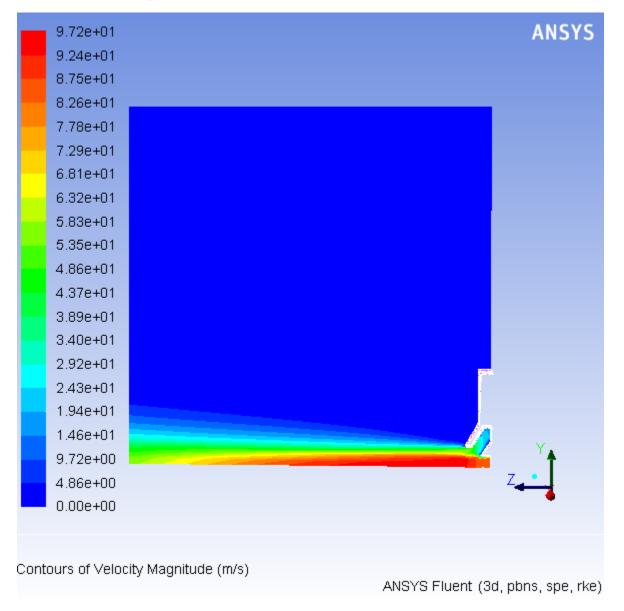


Figure 19.4: Velocity Magnitude at Mid-Point of Atomizer Section

10. Modify the view to include the entire atomizer.

Graphics and Animations → Views...

💶 Views		X
Views back bottom front isometric left right top Save Name view-0	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes =
Apply Came	ra Close	Help

a. Click the **Define...** button to open the **Graphics Periodicity** dialog box.

Graphics Period	icity		— ×
Cell Zones		Associated Surfaces angle = 15 atomizer-wall central_air co-flow-air default-interior fluid	
Periodic Type Translational Rotational Angle (deg) 30 Number of Repeats	Axis Direction X (m) 0 Y (m) 0 Z (m) 1 12	Axis Origin X (m) 0 Y (m) 0 Z (m) 0	
	Set Reset De	fault Close Help	

- 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 19.5: Pathlines of Air in the Swirling Annular Stream (p. 829)).

\bigcirc Graphics and Animations → **\equiv** Pathlines → Set Up...

Pathlines						×
Options	Style			Color by		
Oil Flow	line		•	Particle Variables		•
Reverse Vode Values	A	ttributes		Particle ID		•
Auto Range	Step Size (m)	Tolerance		Min	Max	
Draw Mesh	0.01	0.001		0	92	
Accuracy Control Relative Pathlines	Steps	Path Skip		Release from Surfaces		
XY Plot		5		default-interior		
Write to File	Path Coarsen		٠	fluid		
Туре				outer-wall		
CFD-Post 👻	-			periodic-a		E
	On Zone			periodic-b		
Pulse Mode	atomizer-wall central_air		<u>_</u>	swirling_air		-
Continuous	co-flow-air		=	Highlight Surfaces		
Single	outer-wall			New Surface 🔻		
	periodic-a		Ψ.			
Display	Pulse	Compute Ax	es	Curves Close	Help	

- a. Increase the Path Skip value to 5.
- b. Select swirling_air from the Release from Surfaces selection list.

You will need to scroll down the list to access this item.

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 19.5: Pathlines of Air in the Swirling Annular Stream (p. 829).

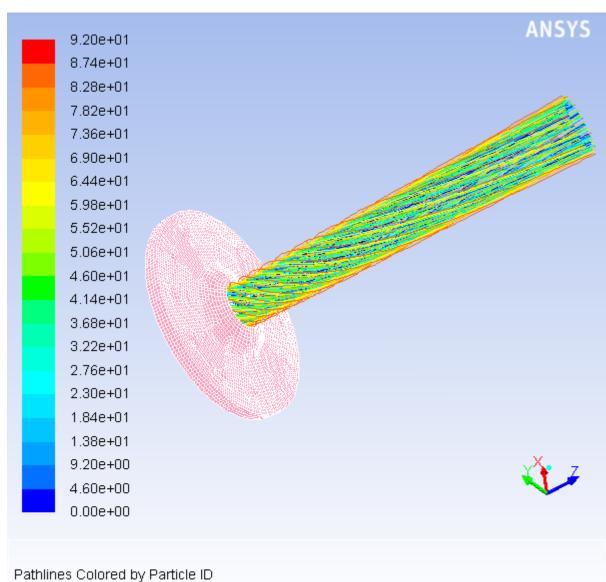


Figure 19.5: Pathlines of Air in the Swirling Annular Stream

ANSYS Fluent (3d, pbns,spe, rke)

19.4.8. Create a Spray Injection

1. Define the discrete phase modeling parameters.

 $\textcircled{} Models \rightarrow \fbox{} Discrete Phase \rightarrow Edit...$

Discrete Phase Model		×
 Discrete Phase Model Interaction Interaction with Continuous Phase Update DPM Sources Every Flow Iteration Number of Continuous Phase Iterations per DPM Iteration Contour Plots for DPM Variables Mean Values RMS Values Tracking Physical Models UDF Numerics Patricking Parameters Max. Number of Steps Spocify Length Scale Step Length Factor Step Length Factor 	Particle Treatment Unsteady Particle Tracking Track with Fluid Flow Time Step Inject Particles at Particle Time Step Index Fluid Flow Time Step Particle Time Step Size (s) 0.0001 Number of Time Steps 10 Image: Clear Particles	
OK Injections DEM Co	llisions 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 Number of Continuous Phase Iterations per DPM Iteration.
- 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 **Tracking** tab, retain the default value of 5 for the **Step Length Factor** tracking option.
- h. Under the Physical Models tab, select the Temperature Dependent Latent Heat.

Discrete Phase Model	
Interaction Interaction with Continuous Phase Update DPM Sources Every Flow Iteration Number of Continuous Phase Iterations per DPM Iteration Contour Plots for DPM Variables Mean Values RMS Values	Particle Treatment Unsteady Particle Tracking Track with Fluid Flow Time Step Inject Particles at Particle Time Step Fluid Flow Time Step Particle Time Step Size (s) Unumber of Time Steps 10 Clear Particles
Tracking Physical Models UDF Numerics Patholic Options Thermophoretic Force Saffman Lift Force Virtual Mass Force Pressure Gradient Force Erosion/Accretion Pressure Dependent Boiling Temperature Dependent Latent Heat Two-Way Turbulence Coupling DEM Collision Stochastic Collision Ø Freakup 	arallel
OK Injections DEM Co	llisions Cancel Help

- i. Select the **Breakup** option in the **Options** group box.
- j. Under the Numerics tab, select Linearize Source Terms.

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.

k. Click Injections... to open the Injections dialog box.

In this step, you will define the characteristics of the atomizer.

Injections	—
Injections	Create
	Сору
	Delete
	List
	Read
	Write
Injection Name Pattern	
Ma	tch
Set Close	elp

I. Click the **Create** button to create the spray injection.

Set Injection Properties					×
Injection Name	Inject	ion Type	Numbe	r of Particle Streams	
injection-0	air-bl	ast-atomizer	▼ 600		
Particle Type				Laws	
Massless	Oroplet	Combusting	Multicomponent	Custom	
Material	Diameter Distribution	Ox	idizing Species	Discrete Phase	Domain
methyl-alcohol-liquid	▼ linear			 none 	•
Evaporating Species	Devolatilizing Species	Pro	duct Species		
ch3oh	•			Ŧ	
Point Properties Physical Mod	els Turbulent Dispersion	Parcel Wet Com	bustion Components	UDF Multiple Reaction	ns ,
Injector Inner Diameter (m)	0.0035	-	<u>^</u>		
Injector Outer Diameter (m)	0.0045				
Spray Half Angle (deg)	-45	constant	•		
Deletion Valentin (m (r)					
Relative Velocity (m/s)	82.6				
Azimuthal Start Angle (deg)	0	1			
Azimuthal Stop Angle (deg)		-			
Aziniotriai Stop Angle (deg)	30		-		
	0	(File C	Help		

i. In the **Set Injection Properties** dialog box, select **air-blast-atomizer** from the **Injection Type** drop-down list.

ii. Enter 600 for Number of Particle Streams.

This option controls the number of droplet parcels that are introduced into the domain at every time step.

- iii. Select **Droplet** in the **Particle Type** group box.
- iv. Select methyl-alcohol-liquid from the Material drop-down list.
- v. In the **Point Properties** tab, retain the default values of 0 and 0 for **X-Position** and **Y-Position**.
- vi. Enter 0.0015 for **Z-Position**.

Scroll down the list to see the remaining point properties.

vii. Retain the default values of 0, 0, and 1 for X-Axis, Y-Axis, and Z-Axis, respectively.

viii.Enter 263 K for Temperature.

ix. 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.

x. 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.

- xi. Enter 0.0035 m for the Injector Inner Diameter and 0.0045 m for the Injector Outer Diameter.
- xii. Enter -45 degrees for Spray Half Angle.

The spray angle is the angle between the liquid sheet trajectory and the injector centerline. In this case, the value is negative because the sheet is initially converging toward the centerline.

xiii.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.

xiv. 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.

xv. Click the **Physical Models** tab to specify the breakup model and drag parameters.

tide Type Laws Massless Inert Diameter Distribution Combusting Multicomponent Custom Laws Custom Custom Custom Custom Control Custom Cu	Article Type Massless Inert Diameter Distribution Combusting Multicomponent Custom Custom Diameter Distribution Oxidizing Species Discrete Phase Domain Iner Product Species Devolatilizing Species Product Species Product Species Product Species Product Species Product Species Data Data Data Data Data Data Data Dat	jection Name	Injection Type	Number of Particle Streams	
Massless Inert Droplet Combusting Multicomponent Custom erial Diameter Distribution Oxidizing Species Discrete Phase Doma thyl-alcohol-liquid Inear Inear Inear Inear porating Species Devolatilizing Species Product Species Discrete Phase Doma porating Species Devolatilizing Species Product Species Inone porating Physical Models Turbulent Dispersion Parcel Wet Combustion Components UDF Multiple Reactions ag Parameters Breakup Inone Inone Inone Inone Inone Orag Law Inone Inone Inone Inone Inone Inone Inone	Massless Inert Droplet Combusting Multicomponent Custom aterial Diameter Distribution Oxidizing Species Discrete Phase Domain nethyl-alcohol-liquid Inear Inear Inear Inear vaporating Species Devolatilizing Species Product Species Discrete Phase Domain h3oh Inear Inear Inear Inear Point Properties Physical Models Turbulent Dispersion Parcel Wet Combustion Components UDF Multiple Reactions Drag Parameters Breakup Breakup Breakup Model Breakup Constants Image: TAB Y0 Image: TAB Image: TAB Y0 Image: TAB Image: TAB Y0 Image: TAB Image: TAB <th>njection-0</th> <th>air-blast-atomizer</th> <th>▼ 600 ●</th> <th></th>	njection-0	air-blast-atomizer	▼ 600 ●	
erial Diameter Distribution Oxidizing Species Discrete Phase Doma thyl-alcohol-liquid Inear Inone none none porating Species Devolatilizing Species Product Species Soh Intervention Physical Models Turbulent Dispersion Parcel Wet Combustion Components UDF Multiple Reactions Breakup ag Parameters Breakup Drag Law Intervention Parcel Breakup Breakup Model Breakup Constants Intervention Parcel Physical Species Product Species Product Species Product Species Product Species Physical Model Preakup Physical Model Breakup Physical	aterial Diameter Distribution Oxidizing Species Discrete Phase Domain nethyl-alcohol-liquid Inear Inone Inear Inone Ino	article Type		Laws	
thyl-alcohol-liquid Inear Inone Ino	ethyl-alcohol-liquid Inear Inone	🔿 Massless 👘 Inert	Droplet Combusting	Multicomponent Custom	
porating Species Devolatilizing Species Product Species Boh	Product Species Product Specie	aterial	Diameter Distribution Oxi	idizing Species Discrete Phase D	omain
Boh Image: Composition of the section of the sectio	h3oh Point Properties Physical Models Turbulent Dispersion Parcel Wet Combustion Components UDF Multiple Reactions Drag Parameters Drag Law dynamic-drag TAB Y0 Wave KHRT Breakup Parcels 2	nethyl-alcohol-liquid 🔹 🔻	linear v	√ none	
oint Properties Physical Models Turbulent Dispersion Parcel Wet Combustion Components UDF Multiple Reactions ag Parameters Breakup Orag Law dynamic-drag TAB y0 0	Point Properties Physical Models Turbulent Dispersion Parcel Wet Combustion Components UDF Multiple Reactions Drag Parameters Breakup Drag Law dynamic-drag TAB Y0 0 Wave KHRT Breakup Parcels 2	aporating Species	Devolatilizing Species Pro	duct Species	
ag Parameters Breakup Orag Law dynamic-drag TAB Breakup Constants 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Drag Parameters Breakup Drag Law dynamic-drag TAB V0 0 Wave KHRT Breakup Parcels 2	h3oh 🔻		Ψ	
dynamic-drag	Drag Law dynamic-drag TAB V0 0 Vave KHRT Breakup Parcels 2	Point Properties Physical Models	Turbulent Dispersion Parcel Wet Comb	bustion Components UDF Multiple Reactions	
dynamic-drag	Drag Law dynamic-drag TAB V0 0 Vave KHRT Breakup Parcels 2	Drag Parameters	Breakup		
dynamic-drag Breakup Model Breakup Constants TAB y0 0	dynamic-drag Breakup Model Breakup Constants TAB V0 Wave KHRT Breakup Parcels T				
TAB y0 0	TAB y0 0 Wave KHRT Breakup Parcels 2			et a	
0	Wave KHRT Breakup Parcels 2	-,			
© Wave	Urconup Forecia /			0	
Unconcept Forces	SSD 🖉		- Di Coltap i di Cela	2	
© SSD ♥			SSD SSD		

xvi.Ensure that Enable Breakup and TAB are enabled in the Breakup group box.

xviiRetain the default values of 0 for **y0** and 2 for **Breakup Parcels** in the **Breakup Constants** group box.

xviiBelect dynamic-drag from the Drag Law drop-down list in the Drag Parameters group box.

The **dynamic-drag** law is available only when the **Breakup** model is used.

xix. Click the **Turbulent Dispersion** tab to define the turbulent dispersion.

Injection Name	Inje	ction Type		Number of Part	
injection-0	air-	-blast-atomizer	•	600	
Particle Type				Laws	
🔿 Massless 👘 Inert	Oroplet	Combusting	Multicompo	nent 📃 O	ustom
Material	Diameter Distributio	in 0	xidizing Species		Discrete Phase Domain
methyl-alcohol-liquid 🔹	linear				none
Evaporating Species	Devolatilizing Specie	es Pr	roduct Species		_
ch3oh 👻		-			·
Point Properties Physical Models	Turbulent Dispersion	n Parcel Wet Cor	mbustion Compor	ents UDF	Multiple Reactions
Stochastic Tracking	Cloud Tracking				
Discrete Random Walk Model	Cloud Model				
☑ Discrete Random Walk Model ☑ Random Eddy Lifetime		iter (m)			
Random Eddy Lifetime	Cloud Model	iter (m)			
Random Eddy Lifetime	Cloud Model Min. Cloud Diame				
Random Eddy Lifetime	Cloud Model Min. Cloud Diame				
Random Eddy Lifetime	Cloud Model Min, Cloud Diame 0 Max, Cloud Diame				
Random Eddy Lifetime Number of Tries 1 Time Scale Constant	Cloud Model Min, Cloud Diame 0 Max, Cloud Diame				
Random Eddy Lifetime Number of Tries 1 Time Scale Constant	Cloud Model Min, Cloud Diame 0 Max, Cloud Diame				
Random Eddy Lifetime Number of Tries 1 Time Scale Constant	Cloud Model Min, Cloud Diame 0 Max, Cloud Diame				
Random Eddy Lifetime Number of Tries 1 Time Scale Constant	Cloud Model Min, Cloud Diame 0 Max, Cloud Diame				
Random Eddy Lifetime Number of Tries 1 Time Scale Constant	Cloud Model Min. Cloud Diame 0 Max. Cloud Diame 100000	eter (m)	Cancel		

xx. 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.

xxi.Click OK to close the Set Injection Properties dialog box.

An **Information** dialog box appears reminding you to confirm the property values before continuing. Click **OK** in the **Information** dialog box to continue.

Note

To modify the existing injection, select its name in the **Injections** list and click **Set...**, or simply double-click the injection of interest.

xxii.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. m. Click **OK** to close the **Discrete Phase Model** dialog box.

2. Specify the droplet material properties.



When secondary atomization models (such as **Breakup**) are used, several droplet properties need to be specified.

Create/Edit Materials		EX
Name	Material Type	Order Materials by
methyl-alcohol-liquid	droplet-particle	Name Chemical Formula
Chemical Formula	Fluent Droplet Particle Materials	Chemical Pormula
ch3oh <l></l>	methyl-alcohol-liquid (ch3oh <l>)</l>	Fluent Database
	Mixture	User-Defined Database
	none 👻	·]
Properties		
Saturation Vapor Pressure (pascal)	piecewise-linear	
Hast of Durchusia (idea)		
Heat of Pyrolysis (j/kg)	constant	
	0	
Vaporization Model		
	convection/diffusion-controlled	
Droplet Surface Tension (n/m)	constant 💌 Edit 🗉	
	0.0222657	
	*	
Chan	ge/Create Delete Close Help	

- 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.
 - i. Review the default values and click OK to close the Piecewise-Linear Profile dialog box.
- e. Select convection/diffusion-controlled from the Vaporisation Model drop-down list.
- f. Click **Change/Create** to accept the change in properties for the methanol droplet material and close the **Create/Edit Materials** dialog box.

19.4.9. Solution

1. Increase the under-relaxation factor for **Discrete Phase Sources**.



Solution Controls	Sol	ution	Controls
-------------------	-----	-------	----------

Pseudo Transient Explicit Relaxation Factors	-
Turbulent Viscosity	-
1	
ch3oh	
0.75	
02	
0.75	
Energy	
0.75	≡
Discrete Phase Sources	
0.9	
	Ŧ
Default	
Equations Limits Advanced	
Set All Species URFs Together	
Help	

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.

 $\textcircled{} Monitors \rightarrow \overleftarrow{\sqsubseteq} Residuals \rightarrow Edit...$

Residual Monitors				
	Equations Residual Monitor Continuity X-velocity Y-velocity Z-velocity			
Iterations to Store	Residual Values Normalize Scale Compute Local Scale	Iterations 5	Convergence Criterion	
OK Plot Renormalize Cancel Help				

- 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**.

Monitors (Surface Monitors) → Create...

Surface Monitor				
Name	Report Type			
surf-mon-1	Mass-Weighted Average 🗸 🗸			
Options	Field Variable			
Print to Console	Species 🔻			
V Plot	Mass fraction of ch3oh 🔹			
Window	Surfaces			
2 Curves Axes	angle=15			
	atomizer-wall			
Write	central_air			
File Name	co-flow-air			
surf-mon-1.out	default-interior			
	fluid			
X Axis	outer-wall			
Iteration 👻	outlet			
Cat Data Sugar	periodic-a			
Get Data Every	periodic-b			
1 Iteration -	swirling_air			
Average Over(Iterations)				
	Highlight Surfaces			
	New Surface 💌			
OK Cancel Help				

- a. Retain **surf-mon-1** for **Name**.
- b. Enable Plot.
- c. Select Mass-Weighted Average from the Report Type drop-down list.
- d. Select Species... and Mass fraction of ch3oh from the Field Variable drop-down lists.
- e. Select **outlet** from the **Surfaces** selection list.
- f. Click **OK** to close the **Surface Monitor** dialog box.
- 4. Enable the plotting of the sum of the **DPM Mass Source**.

 $\clubsuit Monitors (Volume Monitors) \rightarrow Create...$

Volume Monitor	
Name	Report Type
vol-mon-1	Sum
Options	Field Variable
✓ Print to Console	Discrete Phase Sources
V Plot	DPM Mass Source 🔹
Window	Cell Zones
3 Curves Axes	fluid
Write	
File Name	
vol-mon-1.out	
X Axis	
Iteration	
Get Data Every	
1 Iteration	
Average Over	
OK	Cancel Help

- a. Retain vol-mon-1 for Name.
- b. Enable **Plot**.
- c. Click the Axes... button to open the Axes volume Monitor Plot dialog box.

Axes - volume Monitor Plot					
Axis X Y Label	Number Format Type exponential Precision 2	Major Rules			
Options Log Auto Range Major Rules Minor Rules	Range Minimum 0 Maximum 0	Minor Rules Color dark gray Weight 1			
	Apply Close He	lp			

- i. Select **Y** in the **Axis** list.
- ii. Select exponential from the Type drop-down list.
- iii. Set Precision to 2.
- iv. Click Apply and close the Axes volume Monitor Plot dialog box.
- d. Select Sum from the Report Type drop-down list.
- e. Select Discrete Phase Sources... and DPM Mass Source from the Field Variable drop-down lists.
- f. Select **fluid** from the **Cell Zones** selection list.
- g. Click **OK** to close the **Volume Monitor** dialog box.
- 5. Request 200 more iterations (Figure 19.6: Convergence History of Mass Fraction of ch3oh on Fluid (p. 841) and Figure 19.7: Convergence History of DPM Mass Source on Fluid (p. 841)).

CRun Calculation

It can be concluded that the solution is converged because the number of particle tracks are constant and the monitors are flat.

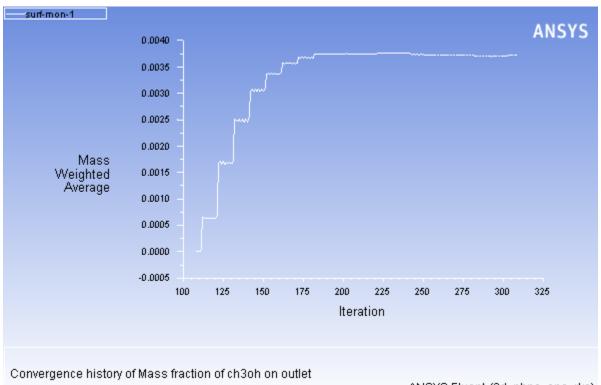


Figure 19.6: Convergence History of Mass Fraction of ch3oh on Fluid

ANSYS Fluent (3d, pbns, spe, rke)

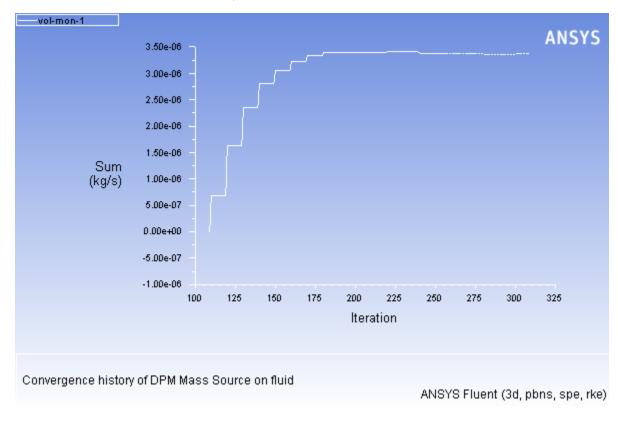


Figure 19.7: Convergence History of DPM Mass Source on Fluid

6. Save the case and data files (spray2.cas.gz and spray2.dat.gz).

File \rightarrow Write \rightarrow Case & Data...

19.4.10. Postprocessing

1. Display the trajectories of the droplets in the spray injection (Figure 19.8: Particle Tracks for the Spray Injection (p. 843)).

This will allow you to review the location of the droplets.

Ç	Gra	ohics and	Animations	→≣	Particle	Tracks →	Set U	p
			/					

Particle Tracks		X		
Options	Track Style	Color by		
Node Values	point -	Particle Variables 🔻		
Auto Range Draw Mesh	Attributes	Particle Diameter		
XY Plot		Min (m) Max (m)		
Write to File	Vector Style	9.68213e-06 8.716901e-05		
	none 🔻	Update Min/Max		
Filter by	Attributes	Track Single Particle Stream		
	Pulse Mode	Stream ID Skip Coarsen		
	 Continuous Single 			
Reporting		Release from Injections		
Report Type Off Summary Current Position	Report to File O Console	injection-0		
Display	Ilse Track	Axes Curves Close Help		

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.

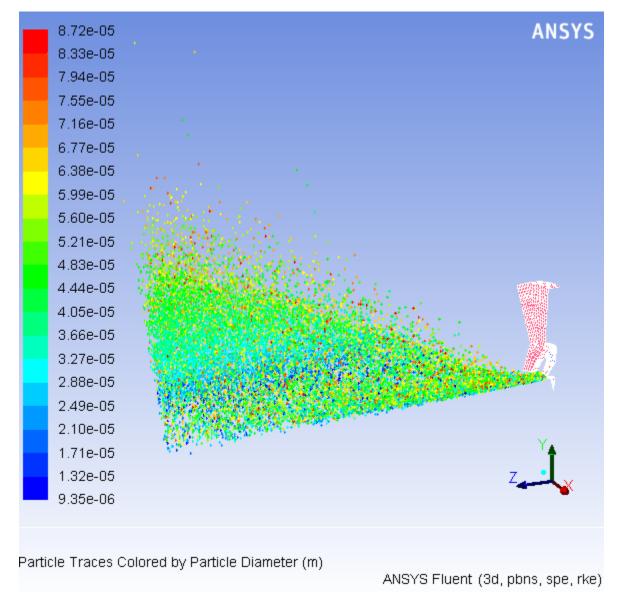
- d. Select injection-0 from the Release from Injections selection list.
- e. Click **Display**. As an optional exercise, you can increase the particle size by clicking the **Attributes...** button in the **Particle Tracks** dialog box and adjusting the **Marker Size** value in the **Track Style Attributes** dialog box.

- f. Close the **Particle Tracks** dialog box.
- g. Restore the 30-degree section to obtain the view as shown in Figure 19.8: Particle Tracks for the Spray Injection (p. 843).

 $\textcircled{} Graphics and Animations \rightarrow Views...$

- i. Click the **Define** button to open **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 19.8: Particle Tracks for the Spray Injection (p. 843).

Figure 19.8: 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 19.9: Contours of DPM Temperature (p. 845)).

Graphics and Anim	mations → 🗮 Con	tours → Set Up		
Contours		×		
Options	Contours of			
✓ Filled	Discrete Phase Variab	Discrete Phase Variables 👻		
Vode Values Global Range	DPM Temperature	•		
Auto Range	Min (k)	Max (k)		
Clip to Range	260	271.8637		
Draw Mesh	Surfaces			
	angle=15	*		
Levels Setup	atomizer-wall	=		
20	central_air co-flow-air			
	default-interior	-		
Surface Name Pattern	New Surface			
Match	Surface Types			
	axis	*		
	clip-surf exhaust-fan			
	fan	-		
Display Compute Close Help				

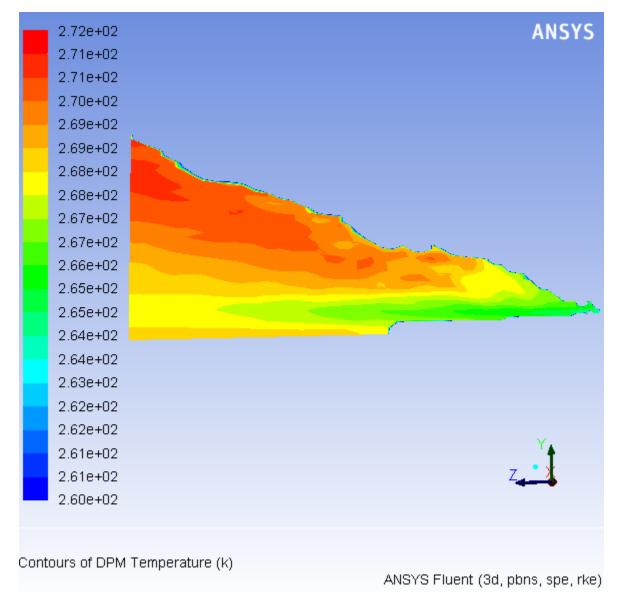
- a. Ensure that Filled is enabled in the Options group box
- b. Disable Draw Mesh.
- c. Select Discrete Phase Variables... and DPM Temperature from the Contours of drop-down lists.

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 19.9: Contours of DPM Temperature (p. 845).

Figure 19.9: Contours of DPM Temperature

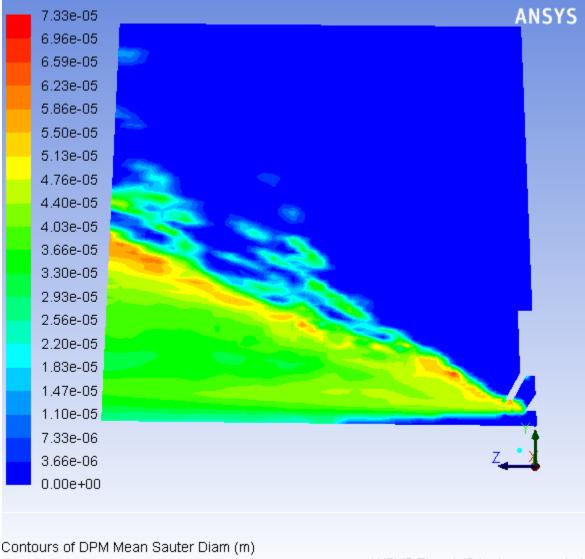


3. Display the mean Sauter diameter (Figure 19.10: Contours of DPM Sauter Diameter (p. 846)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

- a. Enable Filled in the Options group box
- b. Select **Discrete Phase Variables...** and **DPM Mean Sauter Diam** 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.

Figure 19.10: Contours of DPM Sauter Diameter



ANSYS Fluent (3d, pbns, spe, rke)

4. Display vectors of DPM mean velocity colored by DPM velocity magnitude (Figure 19.11: Vectors of DPM Mean Velocity Colored by DPM Velocity Magnitude (p. 848)).



_					
Vectors		— ×			
Options	Vectors of				
Global Range	dpm-mean-velocity	•			
Auto Range	Color by				
Clip to Range Auto Scale	Discrete Phase Variables				
Draw Mesh					
Didw Mesi	DPM Velocity Magnitude	•			
Style		ix (m/s)			
arrow 👻	0 7	9.1813			
Scale Skip	Surfaces				
7 0	angle=15				
	atomizer-wall				
Vector Options	central_air	E			
Custom Vectors	co-flow-air				
Custom vectors	default-interior				
	fluid	-			
Surface Name Pattern					
Match	New Surface 🔻				
	Surface Types				
	axis	*			
	clip-surf				
	exhaust-fan				
	fan	-			
Display	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.

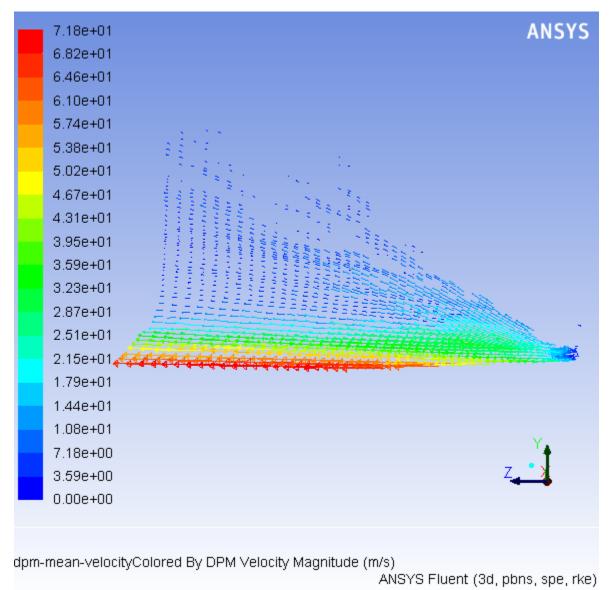


Figure 19.11: Vectors of DPM Mean Velocity Colored by DPM Velocity Magnitude

5. Create an isosurface of the methanol mass fraction.

Surface \rightarrow Iso-Surface...

Iso-Surface		×
Surface of Constant Species Mass fraction of ch3oh Min Max 0 0.009822079 Iso-Values 0.002 Imass-fraction-of-ch3oh-11	From Surface angle = 15 atomizer-wall central_air co-flow-air default-interior fluid . " From Zones fluid	
Create Compute Manage.	Close Help	

- 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**).

$\mathbf{\mathbf{\dot{\mathbf{\nabla}}}} \mathbf{General} \rightarrow \mathbf{Display...}$

💶 Mesh Displa	у		×
Options	Edge Type	Surfaces	
Nodes	All	fluid	
Edges	Feature	methanol-mf=0.002	
Faces	Outline	outer-wall	
Partitions	L	outlet	
Chuicle Calatan	Taabuwa Aaala	periodic-a	=
	Feature Angle	periodic-b	
0	20	swirling_air	~
Surface Name Pa	Match	New Surface Surface Types axis	
		dip-surf	
		exhaust-fan	
		fan -	Ψ.
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			x
Options Color by Type Color by ID Sample	Types interior outlet periodic rans-les-interface symmetry axis wall free-surface internal traction interface surface	▲ E	Colors background black blue cyan dark blue dark gray dark green dark red foreground green light blue light 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.

Graphics and Animations → Views...

a. Click Define... to open the Graphics Periodicity dialog box.

💶 Graphics Period	icity		×
Cell Zones fluid		Associated Surfaces angle=15 atomizer-wall central_air co-flow-air default-interior	
Periodic Type Translational Rotational Angle (deg) 30	Axis Direction X (m) 0 Y (m) 0 Z (m) 1	fluid Axis Origin X (m) Y (m) Z (m)	•
Number of Repeats 12			
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 19.12: Full Atomizer Display with Surface of Constant Methanol Mass Fraction (p. 852).

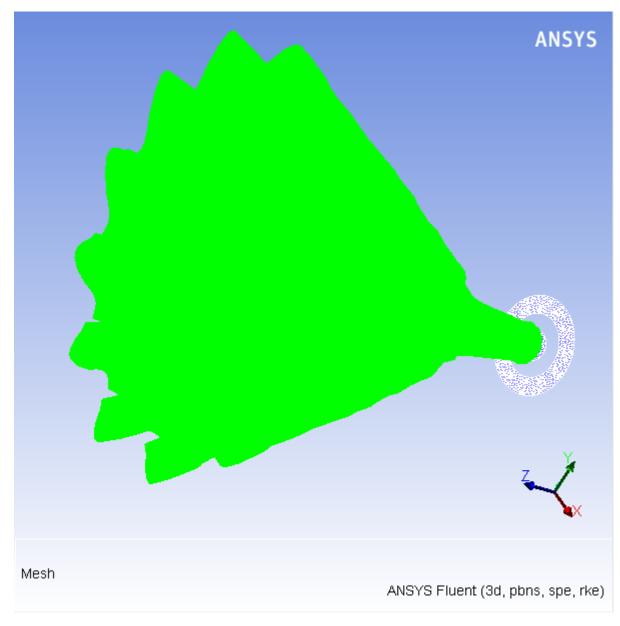


Figure 19.12: Full Atomizer Display with Surface of Constant Methanol Mass Fraction

- e. This view can be improved to resemble Figure 19.13: Atomizer Display with Surface of Constant Methanol Mass Fraction Enhanced (p. 853) 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
 - Change Lighting Method to Flat in the Lights dialog box
 - Check the **Headlight On** check box in the **Lights** dialog box

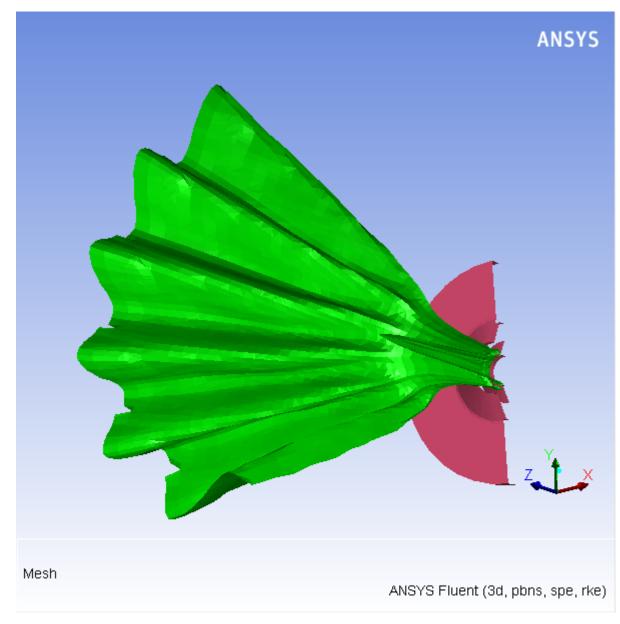


Figure 19.13: Atomizer Display with Surface of Constant Methanol Mass Fraction Enhanced

8. Save the case and data files (spray3.cas.gz and spray3.dat.gz).

 $\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

19.5. 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.

19.6. 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. 123).

Chapter 20: Using the VOF Model

This tutorial is divided into the following sections:

20.1. Introduction20.2. Prerequisites20.3. Problem Description20.4. Setup and Solution20.5. Summary20.6. Further Improvements

20.1. 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.

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. 123)

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

20.3. Problem Description

The problem considers the transient tracking of a liquid-gas interface in the geometry shown in Figure 20.1: Schematic of the Problem (p. 856). 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 20.1: Ink Chamber Dimensions (p. 856).

Figure 20.1: Schematic of the Problem

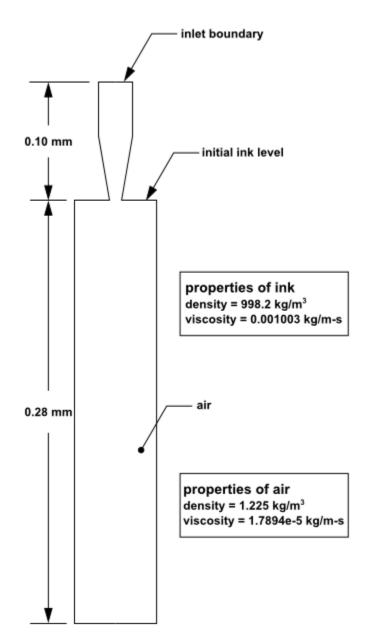


Table 20.1: Ink Chamber Dimensions

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.

20.4. Setup and Solution

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

20.4.1. Preparation 20.4.2. Reading and Manipulating the Mesh 20.4.3. General Settings 20.4.4. Models 20.4.5. Materials 20.4.6. Phases 20.4.7. Operating Conditions 20.4.8. User-Defined Function (UDF) 20.4.9. Boundary Conditions 20.4.10. Solution 20.4.11. Postprocessing

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 15.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip vof_R150.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**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Enable **Double-Precision**.

For more information about Fluent Launcher, see Starting ANSYS Fluent Using 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.

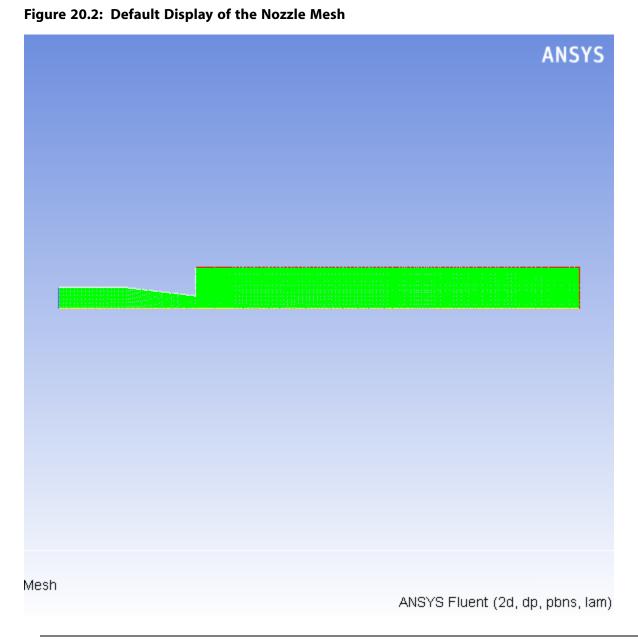
20.4.2. Reading and Manipulating the Mesh

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. 863).

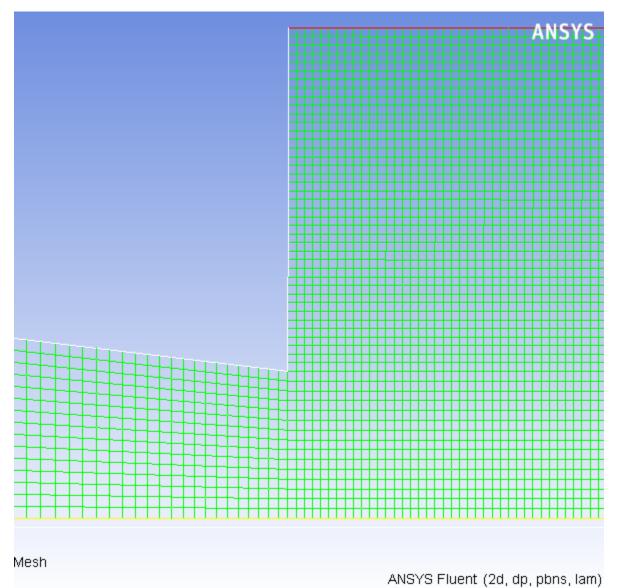
2. Examine the mesh (Figure 20.2: Default Display of the Nozzle Mesh (p. 859)).



Extra

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 20.3: The Quadrilateral Mesh (p. 860)).

Figure 20.3: The Quadrilateral Mesh



3. Set graphics display options

 $\clubsuit Graphics and Animations \rightarrow Options...$

C Display Options	×
Rendering	Graphics Window
Line Width 1 Point Symbol (+)	Active Window Close
Animation Option Wireframe	Color Scheme Workbench 👻
 Double Buffering Outer Face Culling Hidden Line Removal Hidden Surface Removal Removal Method 	Lighting Attributes
Hardware Z-buffer	Layout Ittles Axes Logo Color White Colormap Colormap Alignment Left
Apply Info Lights	Close Help

a. Select All 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.

Graphics and Animations → Views...

Views		×
Views back front Save Name front	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes 🖹 🗐 🗐
Apply Came	ra 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.

Camera Parameters	—		
Camera Position 🗸	Projection perspective		
X (m) 190			
Y (m) 15			
Z (m) 997.4998			
Apply Close Help			

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 20.4: Mesh Display of the Nozzle Mirrored and Upright (p. 863)).

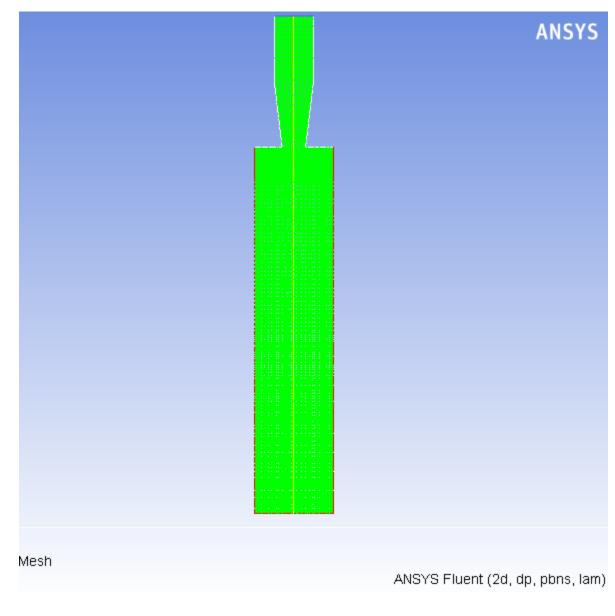


Figure 20.4: Mesh Display of the Nozzle Mirrored and Upright

- ii. Close the Camera Parameters dialog box.
- e. Close the Views dialog box.

20.4.3. General Settings

1. Check the mesh.

$\textcircled{General} \rightarrow Check$

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. Scale the mesh.

```
\textcircled{} General \rightarrow Scale...
```

Scale Mesh		X
Domain Extents		Scaling
Xmin (m) 0 Ymin (m) 0 View Length Unit In m	Xmax (m) 0.0003799999 Ymax (m) 3e-05	Convert Units Specify Scaling Factors Mesh Was Created In Scaling Factors 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.

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.

General → Units...

Set Units			×
Quantities		Units	Set All to
source-specific-dissipation-rate source-temperature-variance source-turbulent-dissipation-rate source-turbulent-viscosity specific-area specific-energy specific-heat stefan-boltzmann-constant surface-density surface-tension		n/m lbf/ft dyn/cm Factor 0.001 Offset 0	default si british cgs
surface-tension-gradient temperature	-	0	
New	List	Close Help	

- 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.

.

General	
Mesh Scale Display	Check Report Quality
Solver	
Type Pressure-Based Density-Based	Velocity Formulation
Time Steady Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

6. Select Transient from the Time list.

7. Select Axisymmetric from the 2D Space list.

20.4.4. Models

1. Enable the Volume of Fluid multiphase model.

```
  \mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Multiphase} \to \mathsf{Edit...}
```

💶 Multiphase Model		×
Model Off Off Volume of Fluid Mixture Eulerian Wet Steam Coupled Level Set + VOF Level Set	Number of Eulerian Phases	
Volume Fraction Parameters Scheme Explicit Implicit Volume Fraction Cutoff 1e-06 Courant Number 0.25 Default Body Force Formulation Implicit Body Force 	Options Open Channel Flow Conal Discretization	
ОК	Cancel Help	

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.

20.4.5. 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.

```
 \diamondsuit Materials \rightarrow \blacksquare air \rightarrow Create/Edit...
```

Name		Material Type	Order Materials by
air		fluid	Name
Chemical Formula		Fluent Fluid Materials	Chemical Formula
		air	✓ Fluent Database
		Mixture	User-Defined Database
		none	-
Properties			
Density (kg/m3)	constant	- Edit	
	1.225		
Viscosity (kg/m-s)	constant	▼ Edit	
	1.7894e-05		
		E	
		÷	

a. Click **Fluent Database...** in the **Create/Edit Materials** dialog box to open the **Fluent Database Ma-terials** dialog box.

Eluent Database Materials	
Fluent Fluid Materials vinyl-silylidene (h2cchsih) vinyl-trichlorosilane (sicl3ch2ch) vinylidene-chloride (ch2ccl2) water-liquid (h2o <l>) water-vapor (h2o) wood-volatiles (wood_vol) (</l>	 Material Type fluid Order Materials by Name Chemical Formula
Properties Density (kg/m3)	Constant View
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 Close 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.

20.4.6. 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.

Phases

Phases
Phases
phase-1 - Primary Phase phase-2 - Secondary Phase
phase-2 - Secondary Phase
Edit Interaction ID 2
Help

1. Specify air (**air**) as the primary phase.

♦ Phases $\rightarrow \stackrel{\frown}{=}$ phase-1 - Primary Phase \rightarrow Edit...

Primary Phase	×
Name	
air	
Phase Material air 🔹 Edit	
OK 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.

Phases →	phase-2	Secondary	Phase \rightarrow	Edit
----------	---------	-----------	---------------------	------

Secondary Phase	×
Name	
water-liquid	
Phase Material water-liquid	
OK 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.

tion
2

Phase Interaction			
Drag Lift Wall Lubrication Turbulent Dispersion	Turbulence Interaction Collisions Slip Heat Mass Reactions	Surface Tension Discretization Interfacial Area	
Surface Tension Force Modeling Model Adhesion Options			
Continuum Surface Force Continuum Surface Stress Jump Adhesion			
Surface Tension Coefficients (dyn/cm)			
water-liquid air	constant v Edit		
	73.5		
	*		
OK Cancel 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.

20.4.7. Operating Conditions

1. Set the operating reference pressure location.

💶 Operati	ing Conditions	X
Pressure		Gravity
	Operating Pressure (pascal) 101325	🔲 Gravity
Reference	Pressure Location	
X (mm)	0.10 P	
Y (mm)	0.03 P	
Z (mm)	0	
]
OK Cancel Help		

Generating Conditions → Operating Conditions...

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.

20.4.8. User-Defined Function (UDF)

1. Interpret the UDF source file for the ink velocity distribution (inlet1.c).

Define \rightarrow **User-Defined** \rightarrow **Functions** \rightarrow **Interpreted...**

Interpreted UDFs	×
Source File Name	
inlet1.c	Browse
CPP Command Name	
фр	
Stack Size	
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.

20.4.9. Boundary Conditions

1. Set the boundary conditions at the inlet (inlet) for the mixture by selecting **mixture** from the **Phase** drop-down list in the **Boundary Conditions** task page.

	it
--	----

💶 Velocity Inlet	×
Zone Name inlet	Phase mixture
Momentum Thermal Radiation Species DPM Multiphase	UDS
Velocity Specification Method Magnitude, Normal to Be	pundary 👻
Reference Frame Absolute	•
Velocity Magnitude (m/s)	udf membrane_speed
Supersonic/Initial Gauge Pressure (pascal)	constant 💌
OK Cancel Help]

- 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.

$\clubsuit Boundary Conditions \rightarrow \blacksquare inlet \rightarrow Edit...$

Velocity Inlet	-X -
Zone Name Inlet	Phase water-liquid
Momentum Thermal Radiation Species DPM Multiphase Volume Fraction 1 constant	uds
OK Cancel Help	

- 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.

 $\diamondsuit Boundary Conditions \rightarrow \blacksquare outlet \rightarrow Edit...$

Pressure Outlet	—
Zone Name outlet	Phase water-liquid
Momentum Thermal Radiation Species DPM Multiphase 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.

 $\textcircled{}{Boundary \ Conditions} \rightarrow \fbox{}{wall_no_wet} \rightarrow \texttt{Edit...}$

-	
🖳 Wall	×
Zone Name	Phase
wall_no_wet	mixture
Adjacent Cell Zone	
fluid	
Momentum Thermal Radiation Species DPM Multiphase	UDS Wall Film
Wall Motion Motion	
Stationary Wall Relative to Adjacent Cell Zone Moving Wall	
Shear Condition	
 No Slip Specified Shear Specularity Coefficient Marangoni Stress Wall Roughness 	
Roughness Height (mm) 0 constant	v
Roughness Constant 0.5 constant	
Wall Adhesion	
Contact Angles (deg)	
water-liquid air 175	constant
OK Cancel H	lelp

- 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.

```
\clubsuit Boundary Conditions \rightarrow \blacksquare wall_wet \rightarrow Edit...
```

💶 Wall		×
Zone Name	Phase	
wall_wet	mixture	
Adjacent Cell Zone		
fluid		
Momentum Thermal Radiation Species	OPM Multiphase UDS Wall Fi	lm
Wall Motion Motion		
Stationary Wall Relative to Adjace	nt Cell Zone	
O Moving Wall		
Shear Condition		
 No Slip Specified Shear 		
Specularity Coefficient		
🔘 Marangoni Stress		
Wall Roughness		
Roughness Height (mm)	constant 👻	
Roughness Constant 0.5	constant v	
0.5		
Wall Adhesion		
Contact Angles (deg)		
water-liquid air	90 constant	
1		Ŧ
OK	Cancel Help	

- a. Retain the default setting of **90** degrees for **Contact Angles**.
- b. Click **OK** to close the **Wall** dialog box.

20.4.10. Solution

1. Set the solution methods.

Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Fractional Step 🔹	
Spatial Discretization	
Gradient	*
Least Squares Cell Based 👻	
Pressure	
PRESTO!	
Momentum	
QUICK 👻	
Volume Fraction	
Compressive 👻	
	Ŧ
Transient Formulation	
First Order Implicit 👻	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
High Order Term Relaxation Options	
Default	
Help	

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.

Monitors →
 Eresiduals → Edit...

Residual Monitors					×
Options	Equations				_
Print to Console	Residual	Monitor Ch	ieck Converge	nce Absolute Criteria	
V Plot	continuity	\checkmark	\checkmark	0.001	
Window 1 Curves Axes	x-velocity		\checkmark	0.001	
Iterations to Plot	y-velocity	V	1	0.001	-
1000	Residual Values				
Iterations to Store	Normalize	I	terations		
1000	Scale				
	Compute Loca	al Scale			
OK Plot	Renormalize	Cance	el H	elp	

- a. Ensure **Plot** is selected in the **Options** group box.
- b. Click **OK** to close the **Residual Monitors** dialog box.
- 3. Initialize the solution using the default initial values.

CSolution Initialization

Solution Initialization	
Initialization Methods O Hybrid Initialization Standard Initialization Compute from	•
Reference Frame ● Relative to Cell Zone ● Absolute	
Initial Values Gauge Pressure (pascal)	
0 Axial Velocity (m/s) 0 Radial Velocity (m/s) 0 water-liquid Volume Fraction 0	T III
Initialize Reset Patch Reset DPM Sources Reset Statistics	
Help	

- a. Retain the default settings for all the parameters and click Initialize.
- 4. Define a register for the ink chamber region.

Adapt \rightarrow Region...

💶 Region Ada	aption	•
Options	Input Coordinates	
InsideOutside	X Min (mm)	X Max (mm)
Shapes	Y Min (mm)	Y Max (mm)
Quad Circle	0	0.03
Cylinder	Z Min (mm)	Z Max (mm)
Manage	0	0
Controls	0	
	Select Point	ts 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.

Extra

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. 175).

- e. Close the Region Adaption dialog box.
- 5. Patch the initial distribution of the secondary phase (water-liquid).

Solution Initialization → Patch...

Patch		X
Reference Frame Relative to Cell Zone Absolute Phase Water-liquid Variable Volume Fraction	Value 1 Use Field Function Field Function	Zones to Patch fluid Registers to Patch E = hexahedron-r0
	Patch Close Help	

- 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.

← Calculation Activities (Autosave Every (Time Steps)) \rightarrow Edit....

🖬 Autosave 🗾
Save Data File Every (Time Steps) 200 💌
Data File Quantities
Save Associated Case Files
 Only if Modified Each Time
File Storage Options
Retain Only the Most Recent Files
Maximum Number of Data Files 0
Only Associated Case Files are Retained
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).

 $\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}$

8. Run the calculation.

Run Calculation

Run Calculation		
Check Case	Preview Mesh Motion	
Time Stepping Method	Time Step Size (s)	
Fixed •	1.0e-8	
Settings	Number of Time Steps	
	3000	
Options		
Extrapolate Variables Data Sampling for Time Statistics Sampling Interval		
1 Sampling Options		
Time Sampled (s)		
	Reporting Interval	
Profile Update Interval		
Data File Quantities	Acoustic Signals	
Calculate		
Help		

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. Retain the default selection of Fixed in the Time Stepping Method drop-down list.
- d. Click Calculate.

The solution will run for 3000 iterations.

20.4.11. Postprocessing

1. Read the data file for the solution after 6 microseconds (inkjet-1-00600.dat.gz).

 $\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Data...}$

2. Display filled contours of water volume fraction after 6 microseconds (Figure 20.5: Contours of Water Volume Fraction After 6 µs (p. 884)).

\rightarrow Graphics and Animations $\rightarrow \equiv$ Contours \rightarrow Set Up	
Contours	
Options	Contours of
 Filled Node Values Global Range Auto Range 	Phases 👻
	Volume fraction 👻
	Phase
Clip to Range	water-liquid 👻
Draw Mesh	Min Max
Levels Setup 20 1	Surfaces
	default-interior
	inlet 🗧
Surface Name Pattern Match	wall_no_wet
	New Surface
	Surface Types
	axis dip-surf
	exhaust-fan
	fan 👻
Display Compute Close Help	

Ē, Φ-.

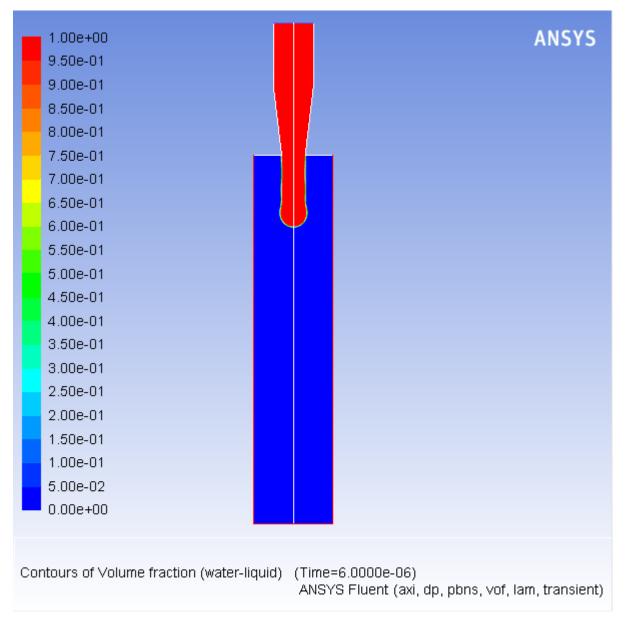
- a. Enable Filled in the Options group box.
- b. Select Phases... and Volume fraction from the Contours of drop-down lists.
- c. Select water-liquid from the Phase drop-down list.
- d. Click **Display**.

Tip

In order to display the contour plot in the graphics window, you may need to click the Fit to Window button.

3. Similarly, display contours of water volume fraction after 12, 18, 24, and 30 microseconds (Figure 20.6: Contours of Water Volume Fraction After 12 µs (p. 885) — Figure 20.9: Contours of Water Volume Fraction After 30 µs (p. 888)).





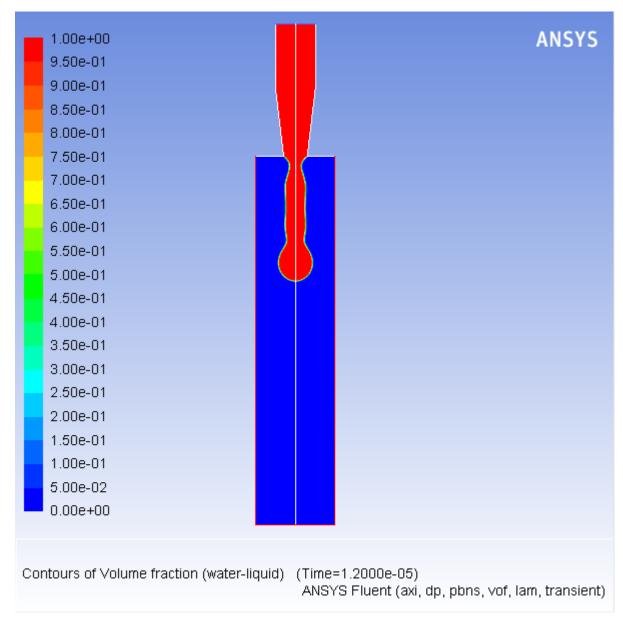
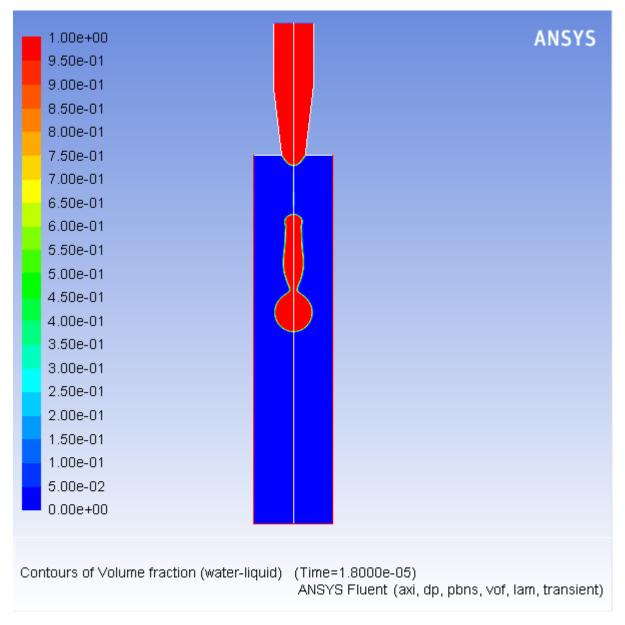


Figure 20.6: Contours of Water Volume Fraction After 12 µs





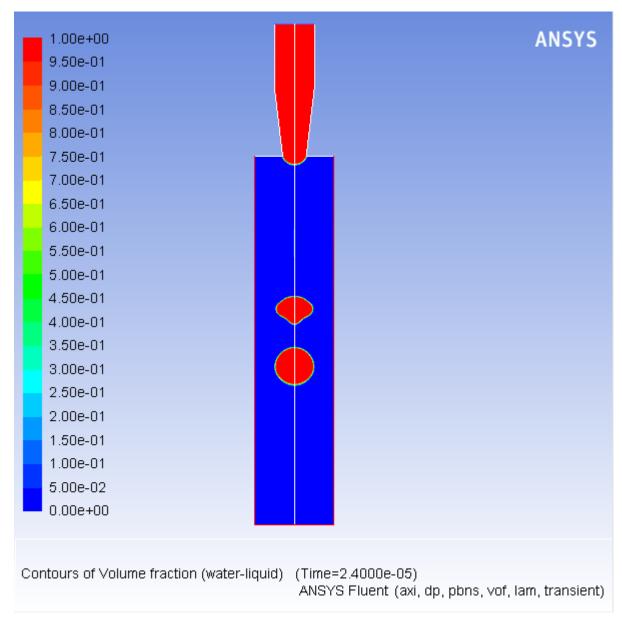
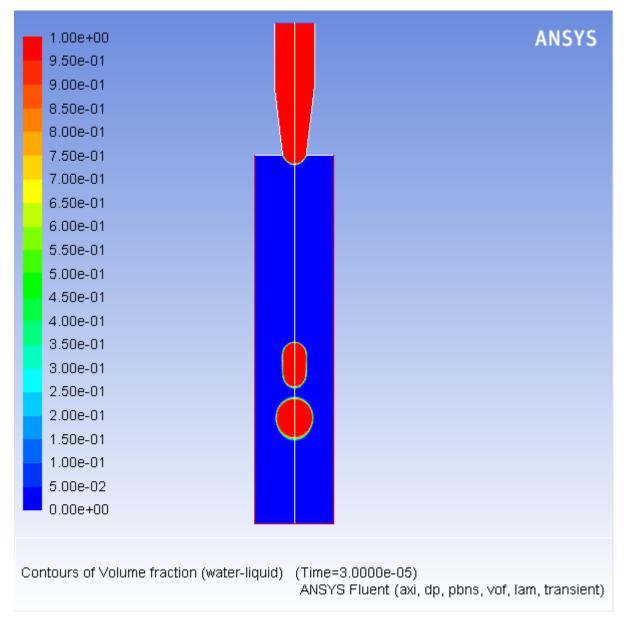


Figure 20.8: Contours of Water Volume Fraction After 24 µs





20.5. 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 Volume of Fluid (VOF) Model Theory of the Theory Guide.

20.6. 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. 123).

Chapter 21: Modeling Cavitation

This tutorial is divided into the following sections:

- 21.1. Introduction
- 21.2. Prerequisites
- 21.3. Problem Description
- 21.4. Setup and Solution
- 21.5. Summary
- 21.6. Further Improvements

21.1. 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.

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. 123)

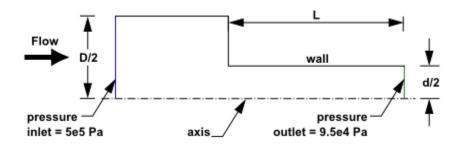
and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

21.3. 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 5×10^5 Pa and an outlet pressure of 9.5×10^4 Pa. The orifice diameter is 4×10^{-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 21.1: Problem Schematic (p. 892).

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. Reading and Checking the Mesh
21.4.3. General Settings
21.4.4. Models
21.4.5. Materials
21.4.6. Phases
21.4.7. Boundary Conditions
21.4.8. Operating 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 15.0 under Version.
- 5. Select this tutorial from the list.

- 6. Click Files to download the input and solution files.
- 7. Unzip the cavitation_R150.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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, 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.

21.4.2. 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.

\bigcirc General \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.

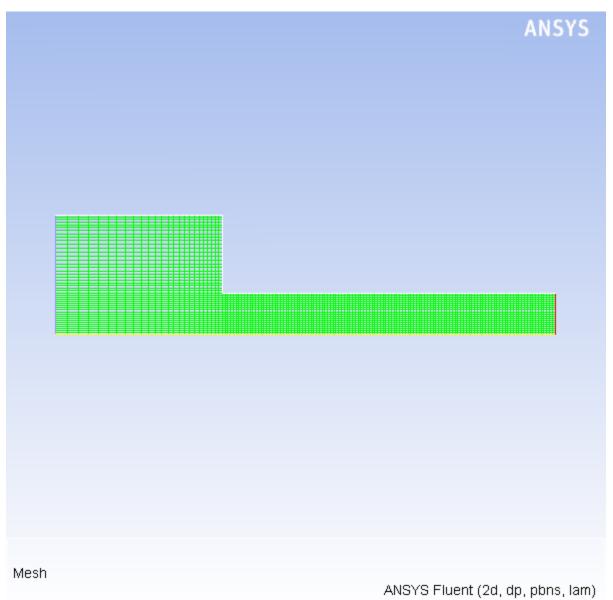
3. Check the mesh scale.

 \bigcirc General \rightarrow Scale...

💶 Scale N	/lesh			×
Domain Ext	tents			Scaling
Xmin (m)	-0.016	Xmax (m)	0.032	 Convert Units Specify Scaling Factors
Ymin (m)	0	Ymax (m)	0.01152	Mesh Was Created In Select>
View Length Unit In m				Scaling Factors X 1 Y 1 Scale Unscale
		C	lose Help	

- a. Retain the default settings.
- b. Close the **Scale Mesh** dialog box.
- 4. Examine the mesh (Figure 21.2: The Mesh in the Orifice (p. 895)).





As seen in Figure 21.2: The Mesh in the Orifice (p. 895), 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.

21.4.3. General Settings

1. Specify an axisymmetric model.

General

General	
Mesh Scale Display	Check Report Quality
Solver	
Type ● Pressure-Based ○ Density-Based	Velocity Formulation
Time ◉ Steady ◯ Transient	2D Space Planar 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.

21.4.4. Models

1. Enable the multiphase mixture model.



Multiphase Model	EX
Model Off Off Volume of Fluid Mixture Eulerian Wet Steam	Number of Eulerian Phases
Mixture Parameters	1
Slip Velocity	
Body Force Formulation	1
Implicit Body Force	
ОКС	ancel 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. 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- ε turbulence model with standard wall functions.

 $\mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Viscous} \to \mathsf{Edit...}$

💶 Viscous Model	
Model	Model Constants C2-Epsilon 1.9 TKE Prandtl Number 1 TDR Prandtl Number 1.2 User-Defined Functions Turbulent Viscosity none
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.

21.4.5. Materials

1. Create a new material to be used for the primary phase.

 $\diamondsuit Materials \rightarrow \blacksquare Fluid \rightarrow Create/Edit...$

Create/Edit Mat	erials		×
Name water		Material Type	Order Materials by ● Name
Chemical Formula		Fluent Fluid Materials water	Chemical Formula Fluent Database User-Defined Database
Properties		Mixture none	
Density (kg/m3)	constant 1000	Edit	
Viscosity (kg/m-s)	constant 0.001	✓ Edit	
	Change/Create	Delete Close Help	

- a. Enter water for Name.
- b. Enter 1000 kg/m³ 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		٢			
Fluent Fluid Materials vinyl-silylidene (h2cchsih) vinyl-trichlorosilane (sid3ch2ch) vinylidene-chloride (ch2ccl2) water-liquid (h2o <l>) water-vapor (h2o) wood-volatiles (wood_vol) <</l>	 Material Type fluid Order Materials by Name Chemical Formula 				
Properties Density (kg/m3) Constant View 0.5542 Cp (Specific Heat) (j/kg-k) piecewise-polynomial View					
Thermal Conductivity (w/m-k)					
Viscosity (kg/m-s)	view I.34e-05	Ŧ			
New	Save Copy Close 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 Type	Order Materials by
water-vapor		fluid	 Name
Chemical Formula		Fluent Fluid Materials	Chemical Formula
h2o		water-vapor (h2o)	← Fluent Database
		Mixture	User-Defined Database
		none	-
Properties			
Density (kg/m3)	constant	Edit	
	0.02558		
Viscosity (kg/m-s)	constant	▼ Edit	
	1.26e-06		
		E	
		-	
	Change/Cre	ate Delete Close	Help

- b. Enter 0.02558 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.

21.4.6. Phases

⇔Phases

Phases

Phases	
phase-1 - Primary Phase	
phase-2 - Secondary Phase	
L	
Edit Interaction	ID 2
Help	

1. Specify liquid water as the primary phase.



Primary Phase	×
Name	
liquid	
Phase Material water	
OK Cancel Help	

- a. Enter liquid for Name.
- b. Retain the default selection of water from the Phase Material drop-down list.
- c. Click OK to close the Primary Phase dialog box.
- 2. Specify water vapor as the secondary phase.

 $\diamondsuit \mathsf{Phases} \to \overleftarrow{\sqsubseteq} \mathsf{phase-2} \to \mathsf{Edit...}$

Secondary Phase	×
Name	
vapor	
Phase Material water-vapor	
OK Cancel Help	

- a. Enter vapor for Name.
- b. Select water-vapor from the Phase Material drop-down list.
- c. Click **OK** to close the **Secondary Phase** dialog box.
- 3. Enable the cavitation model.

\mathbf{P} Phases \rightarrow Interaction...

2 P	hase	Interact	ion									.
Dr	g	uft	Wall Lubrication	Furbulent Dispers	on Turbulen	ce Interaction Collisions	Slip H	eat Mas	Reactions	Surface Tension	Discretization	Interfacial Area
			ransfer Mechanism									
	ss Tra	nsfer										
		From Phase		To Phase		Mechanism		ŕ	î			
		liquid		vapor	-	cavitation	•	Edit				
								- H				
-								_				
						OK Can	Help					

a. Click the **Mass** tab.

The dialog box expands to show relevant modeling parameters.

i. Set Number of Mass Transfer Mechanisms to 1.

Click **OK** in the dialog box that appears to inform you that Linearized-Mass-Transfer UDF is on.

- ii. Ensure that **liquid** is selected from the **From Phase** drop-down list in the **Mass Transfer** group box.
- iii. Select vapor from the To Phase drop-down list.
- iv. Select cavitation from the Mechanism drop-down list.

The **Cavitation Model** dialog box will open to show the cavitation inputs.

Cavitation Model	—
Model Schnerr-Sauer Zwart-Gerber-Belamri Cavitation Properties Vaporization Pressure (pascal) constant Edit 3540	Model Constants Bubble Number Density 1e+13
OK Cancel Help	

- A. Retain the default settings.
- B. Retain the value of 3540 Pa for Vaporization Pressure.

The vaporization pressure is a property of the working liquid, which depends mainly on the liquid temperature. The default value is the vaporization pressure of water at a temperature of 300 K.

- C. Click OK to close the Cavitation Model dialog box.
- b. Click **OK** to close the **Phase Interaction** dialog box.

21.4.7. 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.

Order Boundary Conditions

Boundary Conditions	
Zone	
default-interior	
inlet_1	
inlet_2 outlet	
symm_1	
symm_2	
wall	
L	
Phase Type ID	
mixture	
Edit Copy Profiles	
Parameters Operating Conditions	
Display Mesh Periodic Conditions	
Help	
Theip	

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.

 $\clubsuit Boundary Conditions \rightarrow \blacksquare inlet_1 \rightarrow Edit...$

Pressure Inlet	.
Zone Name	Phase
inlet_1	mixture
Momentum Thermal Radiation Species DPM Multiphase	UDS
Reference Frame Absolute	▼
Gauge Total Pressure (pascal) 500000	constant 🔻
Supersonic/Initial Gauge Pressure (pascal) 449000	constant 🔻
Direction Specification Method Normal to Boundary	•
Turbulence	
Specification Method K and Epsilon	•
Turbulent Kinetic Energy (m2/s2)	constant 🔹
Turbulent Dissipation Rate (m2/s3)	constant 🗸
OK Cancel Help	

- a. Select **vapor** from the **Phase** drop-down list.
- b. Enter 500000 Pa for Gauge Total Pressure.
- c. Enter 449000 Pa.

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.

- d. Retain the default selection of **Normal to Boundary** from the **Direction Specification Method** dropdown list.
- e. Select K and Epsilon from the Specification Method drop-down list in the Turbulence group box.
- f. Enter $0.02 \text{ m}^2/\text{s}^2$ for **Turbulent Kinetic Energy**.
- g. Retain the value of $1 \text{ m}^2/\text{s}^3$ for **Turbulent Dissipation Rate**.
- h. 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 Inlet	x	
Zone Name inlet_1	Phase vapor	
Momentum Thermal Radiation Species DPM Multiphase Volume Fraction 0 constant		
OK Cancel Help		

- i. Click the **Multiphase** tab and 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	—
From Boundary Zone inlet_1 inlet_2 symm_1 symm_2	To Boundary Zones 🕃 🗐 🚍
Phase mixture]
Сору	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.

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{=} $ outlet \rightarrow Edit
--

Pressure Outlet	—
Zone Name	Phase
outlet	mixture
Momentum Thermal Radiation Species DPM Multiphase	UDS
Gauge Pressure (pascal) 95000	constant 🔹
Backflow Direction Specification Method Normal to Boundar	ry 🔹
Turbulence	
Specification Method K and Epsilon	▼
Backflow Turbulent Kinetic Energy (m2/s2)	constant 🔻
Backflow Turbulent Dissipation Rate (m2/s3)	⊂ constant
	·
OK Cancel Hel	p

- 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 $0.02 \text{ m}^2/\text{s}^2$ 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.

$\clubsuit Boundary \ Conditions \rightarrow \overleftarrow{\sqsubseteq} outlet$

- a. Select vapor from the Phase drop-down list.
- b. Click Edit... to open the Pressure Outlet dialog box.

Pressure Outlet	—	
Zone Name outlet	Phase vapor	
Momentum Thermal Radiation Species DPM Multiph Backflow Volume Fraction 0 constant		
OK Cancel Help		

- i. Click the **Multiphase** tab and retain the default value of 0 for **Volume Fraction**.
- ii. Click **OK** to close the **Pressure Outlet** dialog box.

21.4.8. Operating Conditions

1. Set the operating pressure.

Boundary Conditions → Operating Conditions...

Operating Conditions	— ×
Pressure	Gravity
Operating Pressure (pascal)	Gravity
O	
Reference Pressure Location	
X (m) 0 P	
Y (m) 0	
Z (m)	
OK Cancel Help	,

- a. Enter 0 Pa for **Operating Pressure**.
- b. Click OK to close the Operating Conditions dialog box.

21.4.9. Solution

1. Set the solution parameters.

CSolution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Coupled with Volume Fractions	
Spatial Discretization	
Pressure	*
PRESTO! -	
Momentum	—
QUICK 👻	
Volume Fraction	
QUICK 🗸	=
Turbulent Kinetic Energy	_
First Order Upwind 👻	
Turbulent Dissipation Rate	
First Order Upwind 👻	-
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
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 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.

Solution Controls

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Pressure	.
0.5	
Momentum	
0.5	E
Density	
1	
Body Forces	
1	
Volume Fraction	
0.5	÷
Default	
Equations Limits Advanced	
Help	

a. Retain the default values in the **Pseudo Transient Explicit Relaxation Factors** group box.

3. Enable the plotting of residuals during the calculation.

Residual Monitors					×
Options	Equations				
▼ Print to Console	continuity	V		3e-07	^
Vindow	x-velocity			1e-05	
1 Curves Axes	y-velocity	V		1e-05	E
Iterations to Plot	k	V		1e-05	
1000	epsilon	V	V	1e-05	-
	Residual Values			Convergence Cr	iterion
Iterations to Store	Normalize		Iterations	absolute	-
1000			5		
	Scale				
	Compute Loca	l Scale			
OK Plot Renormalize Cancel Help					

 $\textcircled{} Monitors \rightarrow \overleftarrow{} Residuals \rightarrow Edit...$

a. Ensure that **Plot** is enabled in the **Options** group box.

- b. Enter 3e-07 for the Absolute Criteria of continuity.
- c. Enter 1e-05 for the Absolute Criteria of x-velocity, y-velocity, k, and epsilon.

Decreasing the criteria for these residuals will improve the accuracy of the solution.

- d. Click OK to close the Residual Monitors dialog box.
- 4. Initialize the solution.

Solution Initialization			
Solution Initialization			
Initialization Methods			
 Hybrid Initialization Standard Initialization 			
More Settings Initialize			
Patch			
Reset DPM Sources Reset Statistics			
Help			

- a. Select Hybrid Initialization from the Initialization Methods group box.
- b. Click More Settings... to open the Hybrid Initialization dialog box.

Hybrid Initialization		
General Settings Turbulence Settings Species Settings		
Number of Iterations 10		
Explicit Under-Relaxation Factor		
Scalar Equation-0		
Scalar Equation-1		
Reference Frame		
 Relative to Cell Zone Absolute 		
Initialization Options		
Use Specified Initial Pressure on Inlets Use External-Aero Favorable Settings Maintain Constant Velocity Magnitude		
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 Steps in Using 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).

```
File \rightarrow Write \rightarrow Case...
```

6. Start the calculation by requesting 400 iterations.

Run Calculation

Run Calculation	
Check Case Preview Mesh Motion.	
Pseudo Transient Options	
Fluid Zone	
Time Step Method Timescale Factor	
O User Specified 1	
Length Scale Method Verbosity	
Conservative	
Number of Iterations Reporting Interval	
Profile Update Interval	
Data File Quantities Acoustic Signals	
Calculate	
Help	

a. Enter 400 for Number of Iterations.

b. Click Calculate.

The solution will converge in approximately 300 iterations.

7. Save the data file (cav.dat.gz).

```
File \rightarrow Write \rightarrow Data...
```

21.4.10. Postprocessing

1. Plot the pressure in the orifice (Figure 21.3: Contours of Static Pressure (p. 915)).

Graphics and <i>I</i>	Anin	nations → 🗮 Con	tours $ ightarrow$ Set Up		
Contours			×		
Options		Contours of			
 ✓ Filled ✓ Node Values ✓ Global Range 		Pressure 👻			
		Static Pressure			
Auto Range		Phase			
Clip to Range		mixture 👻			
Draw Profiles		Min (pascal)	Max (pascal)		
Draw Mesh	3540	499069.3			
Levels Setup	_	Surfaces			
20 🔺 1		default-interior	A		
	S	inlet_1	=		
		inlet_2 outlet			
Surface Name Pattern		symm 1	-		
Ma	atch	[/]	*		
		New Surface 💌			
		Surface Types			
		axis	*		
		clip-surf			
		exhaust-fan			
		fan -	•		
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** and close the **Contours** dialog box.

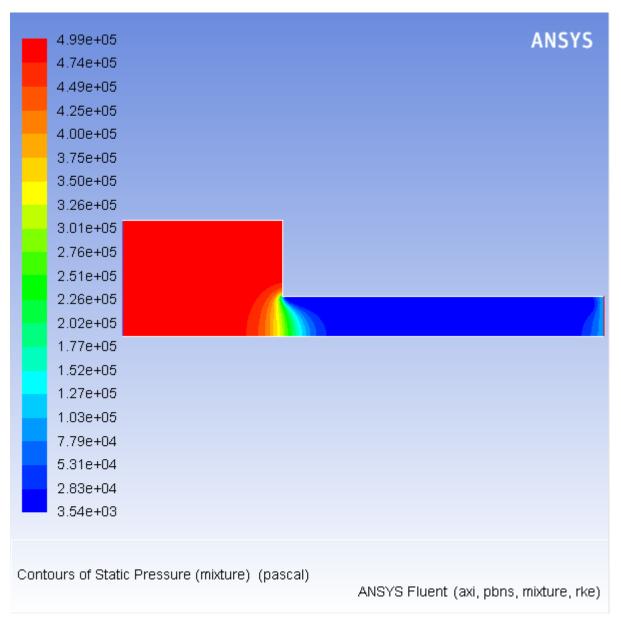


Figure 21.3: Contours of Static Pressure

Note the dramatic pressure drop at the flow restriction in Figure 21.3: Contours of Static Pressure (p. 915). Low static pressure is the major factor causing cavitation. Additionally, turbulence contributes to cavitation due to the effect of pressure fluctuation (Figure 21.4: Mirrored View of Contours of Static Pressure (p. 917)) and turbulent diffusion (Figure 21.5: Contours of Turbulent Kinetic Energy (p. 918)).

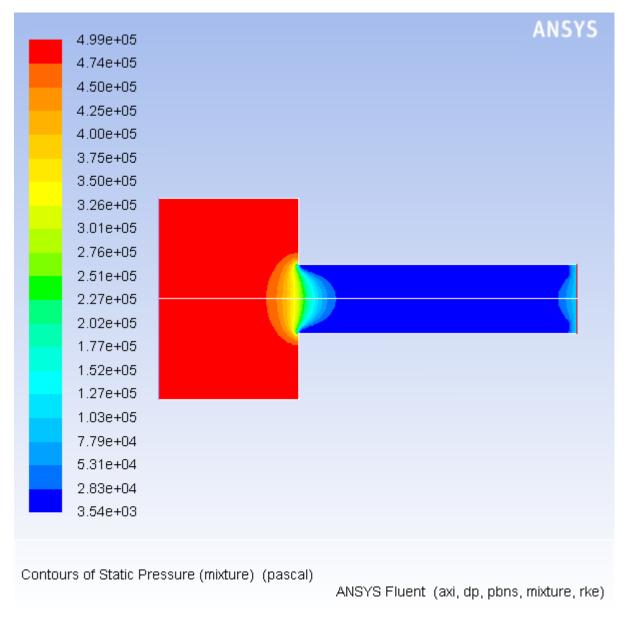
2. Mirror the display across the centerline (Figure 21.4: Mirrored View of Contours of Static Pressure (p. 917)).

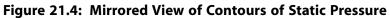
Graphics and Animations \rightarrow Views...

Mirroring the display across the centerline gives a more realistic view.

Views		×		
Views back front Save Name view-0	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes 🖹 🖹 🗐		
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.





3. Plot the turbulent kinetic energy (Figure 21.5: Contours of Turbulent Kinetic Energy (p. 918)).

Graphics and Animations → \blacksquare Contours → Set Up...

- a. Select Turbulence... and Turbulent Kinetic Energy(k) from the Contours of drop-down lists.
- b. Click **Display**.

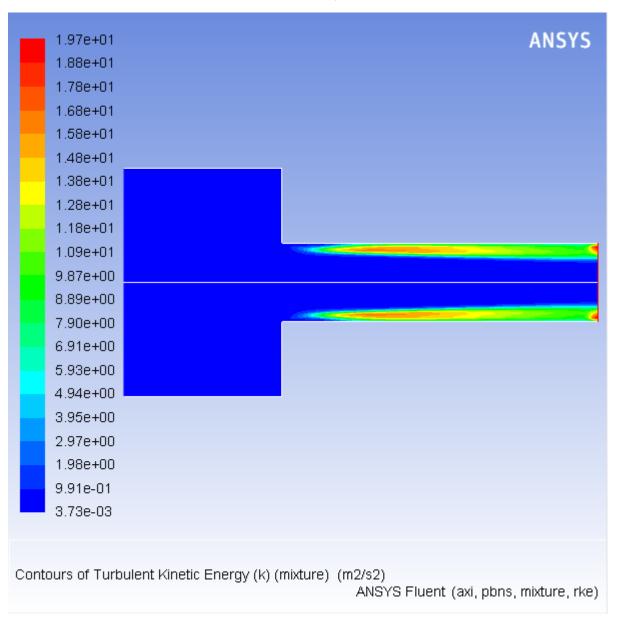


Figure 21.5: Contours of Turbulent Kinetic Energy

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. Plot the volume fraction of water vapor (Figure 21.6: Contours of Vapor Volume Fraction (p. 919)).

Graphics and Animations → \equiv Contours → Set Up...

- a. Select Phases... and Volume fraction from the Contours of drop-down lists.
- b. Select **vapor** from the **Phase** drop-down list.
- c. Click **Display** and close the **Contours** dialog box.

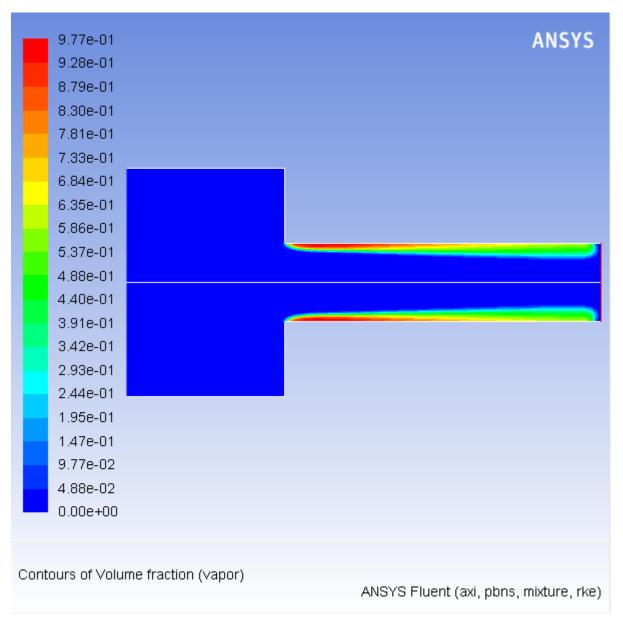


Figure 21.6: Contours of Vapor Volume Fraction

The high turbulent kinetic energy region near the neck of the orifice in Figure 21.5: Contours of Turbulent Kinetic Energy (p. 918) coincides with the highest volume fraction of vapor in Figure 21.6: Contours of Vapor Volume Fraction (p. 919). This indicates the correct prediction of a localized high phase change rate. The vapor then gets convected downstream by the main flow.

21.5. 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.

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. 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. 123).

Chapter 22: Using the Mixture and Eulerian Multiphase Models

This tutorial is divided into the following sections:

22.1. Introduction22.2. Prerequisites22.3. Problem Description22.4. Setup and Solution22.5. Summary22.6. Further Improvements

22.1. 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.

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:

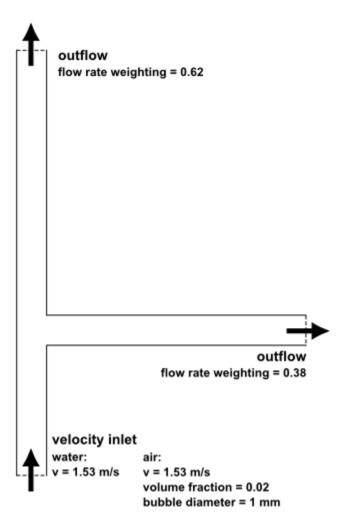
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

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

22.3. 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 22.1: Problem Specification (p. 922).

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. Mesh
22.4.3. General Settings
22.4.4. Models
22.4.5. Materials
22.4.6. Phases
22.4.7. Boundary Conditions
22.4.8. Operating Conditions
22.4.9. Solution Using the Mixture Model
22.4.10. Postprocessing for the Mixture Solution
22.4.11. Higher Order Solution using the Mixture Model

22.4.12. Setup and Solution for the Eulerian Model 22.4.13. Postprocessing for the Eulerian Model

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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip mix_eulerian_multiphase_R150.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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, 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.

22.4.2. Mesh

1. Read the mesh file tee.msh.

$\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

As ANSYS Fluent reads the mesh file, it will report the progress in the console.

22.4.3. General Settings

1. Check the mesh.

$\mathbf{O}_{General} \rightarrow \mathbf{Check}$

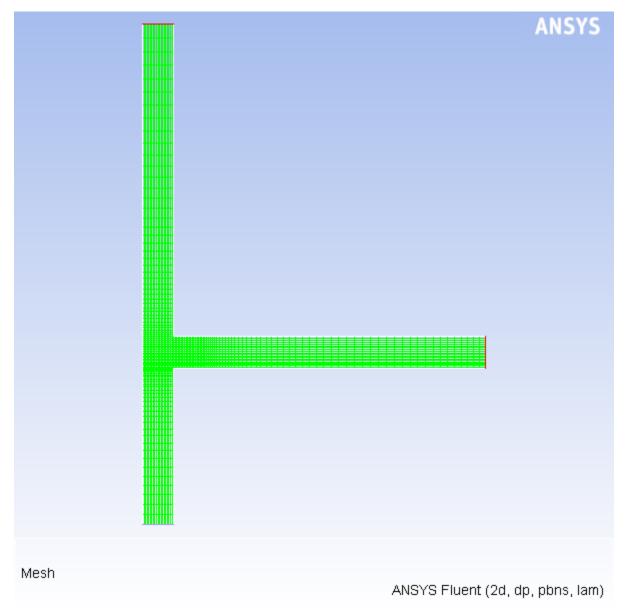
ANSYS Fluent will perform various checks on the mesh and will report the progress in the console. Ensure that the reported minimum volume is a positive number.

2. Examine the mesh (Figure 22.2: Mesh Display (p. 925)).

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 22.2: Mesh Display



3. Retain the default settings for the pressure-based solver.



General	
Mesh Scale Display	Check Report Quality
Solver	
Type Pressure-Based Density-Based	Velocity Formulation
Time Steady Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

22.4.4. Models

1. Select the mixture multiphase model with slip velocities.

 $\textcircled{Models} \rightarrow \overleftarrow{\sqsubseteq} Multiphase \rightarrow Edit...$

a. Select Mixture in the Model list.

The **Multiphase Model** dialog box will expand to show the inputs for the mixture model.

Multiphase Model	—
Model Off Volume of Fluid Mixture Eulerian Wet Steam	Number of Eulerian Phases
Mixture Parameters Image: Slip Velocity	
Body Force Formulation]
ОКС	ancel 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.

 \bigcirc Models $\rightarrow \equiv$ Viscous \rightarrow Edit...

2. Select the realizable k- ε turbulence model with standard wall functions.

Model	Model Constants
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) c-epsilon Model Standard RNG RNG Standard Wall Functions Scalable Wall Functions Scalable Wall Functions Scalable Wall Functions Enhanced Wall Treatment User-Defined Wall Functions Coptions Curvature Correction Production Limiter Mixture Drift Force	C2-Epsilon TKE Prandtl Number 1 TDR Prandtl Number 1.2 Dispersion Prandtl Number 0.75 User-Defined Functions Turbulent Viscosity none

- a. Select **k-epsilon** in the **Model** list.
- b. Select Realizable under in the k-epsilon Model list.

The realizable k- ε 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.

22.4.5. Materials

1. Copy the properties for liquid water from the materials database so that it can be used for the primary phase.

 $\textcircled{} Materials \rightarrow \blacksquare Fluid \rightarrow Create/Edit...$

a. Click the Fluent Database... button to open the Fluent Database Materials dialog box.

Fluent Database Materials		×
Fluent Fluid Materials Vinyl-silylidene (h2cchsih) vinyl-trichlorosilane (sid3ch2ch) vinylidene-chloride (ch2cd2) water-liquid (h2o <l>) water-vapor (h2o) wood-volatiles (wood_vol) < Copy Materials from Case Delete</l>		Material Type fluid Order Materials by Name Chemical Formula
Density (kg/m3) Cp (Specific Heat) (j/kg-k)	constant 998.2 constant 4182	✓ View E
Thermal Conductivity (w/m-k) Viscosity (kg/m-s)	constant 0.6 constant 0.001003	 ▼ View ▼ View
New Edit	Save	Copy Close 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.

22.4.6. Phases

In the following steps you will define the liquid water and air phases that flow in the tee junction.

Phases	
Phases	
phase-1 - Primary Phase phase-2 - Secondary Phase	
Edit Interaction	ID 2
Help	

1. Specify liquid water as the primary phase.

Primary Phase	×
Name	
water	
Phase Material water-liquid	
OK 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.

💶 Secondary I	Phase	×
Name		
air		
Phase Material	air 🔻 Edit	
🔲 Granular		
	rea Concentration	
Properties		
Diameter (m)	constant	Â
	0.001	
		~
	OK Cancel Help	

- 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.

 \mathbf{P} Phases \rightarrow Interaction...

		ispersion Collisions Slip Heat Mass Reactions Surface Tensio	on Discretization Interfacial Area
Drag Coefficient	water	schiler-naumann v Edit	
OK Cancel Help			

a. 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.

b. Click **OK** to close the **Phase Interaction** dialog box.

22.4.7. Boundary Conditions

Boundary Con	ditions	
Zone		
default-interior		
outflow-3 outflow-5		
velocity-inlet-4		
wall-1		
Phase	Туре	ID
air	✓ velocity-inlet ✓	4
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.

$\textcircled{P} Boundary \ Conditions \rightarrow \overleftarrow{E} velocity-inlet-4 \rightarrow Edit...$

Velocity Inlet	—
Zone Name	Phase
velocity-inlet-4	mixture
Momentum Thermal Radiation Species DPM Multiphase	UDS]
Supersonic/Initial Gauge Pressure (pascal)	constant 💌
Turbulence	
Specification Method Intensity and Length Sca	le 🔻
Turbulent Intensity	(%) 5 P
Turbulent Length Scale	e (m) 0.025
OK Cancel Help]

- a. Select Intensity and Length Scale from the Specification Method drop-down list.
- b. Retain the default of 5 % for **Turbulent Intensity**.
- c. Enter 0.025 m for **Turbulent Length Scale**.
- 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).

Goundary Conditions → Evelocity-inlet-4

- a. Select water from the Phase drop-down list.
- b. Click Edit... to open the Velocity Inlet dialog box.

💶 Velocity Inlet	×
Zone Name velocity-inlet-4	Phase water
Momentum Thermal Radia Velocity Specification Method	tion Species DPM Multiphase UDS Magnitude, Normal to Boundary
Reference Frame Velocity Magnitude (m/s)	Absolute
	OK Cancel Help

- i. Retain the default selection of **Magnitude**, **Normal to Boundary** from the **Velocity Specification Method** drop-down list.
- ii. Retain the default selection of Absolute from the Reference Frame drop-down list.
- iii. Enter 1.53 m/s for Velocity Magnitude.
- iv. Click OK to close the Velocity Inlet dialog box.
- 3. Set the boundary conditions at the velocity inlet (velocity-inlet-4) for the secondary phase (air).

◆Boundary Conditions → 🔄 velocity-inlet-4

- a. Select air from the Phase drop-down list.
- b. Click **Edit...** to open the **Velocity Inlet** dialog box.
 - i. Retain the default selection of **Magnitude**, **Normal to Boundary** from the **Velocity Specification Method** drop-down list.
 - ii. Retain the default selection of Absolute from the Reference Frame drop-down list.
 - iii. 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.

iv. Click the Multiphase tab and enter 0.02 for Volume Fraction.

💶 Velocity Inlet	×
Zone Name velocity-inlet-4	Phase air
Momentum Thermal Radiation Species DPM Mul Volume Fraction 0.02 constant	Itiphase UDS
OK Cancel Hel	lp

- v. Click **OK** to close the **Velocity Inlet** dialog box.
- 4. Set the boundary conditions at **outflow-5** for the mixture.

◆Boundary Conditions → Èoutflow-5

- a. Select **mixture** from the **Phase** drop-down list.
- b. 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.

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{=}$ outflow-3 \rightarrow Edit...

- a. Enter 0.38 for Flow Rate Weighting.
- b. Click **OK** to close the **Outflow** dialog box.

22.4.8. Operating Conditions

1. Set the gravitational acceleration.

 $\textcircled{P} Boundary \ Conditions \rightarrow Operating \ Conditions...$

💶 Opera	ting Conditions	×		
Pressure		Gravity		
Referen X (m) Y (m) Z (m)	Operating Pressure (pascal) 101325 P ce Pressure Location P 0 P 0 P 0 P 0 P	♥ Gravity Gravitational Acceleration X (m/s2) 0 Y (m/s2) -9.81 Z (m/s2) 0 Variable-Density Parameters Variable-Density Parameters Image: Specified Operating Density Operating Density (kg/m3) 0		
OK Cancel Help				

a. Enable Gravity.

The **Operating Conditions** dialog box will expand to show additional inputs.

- b. Enter -9.81 m/s^2 for **Y** in the **Gravitational Acceleration** group box.
- c. Enable Specified Operating Density.
- d. Enter 0 kg/m³ 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 Operating Density in the Fluent User's Guide.

e. Click OK to close the Operating Conditions dialog box.

22.4.9. 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.

Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	
Least Squares Cell Based 💌	
Pressure	
PRESTO!	
Momentum	
First Order Upwind	L
Volume Fraction	
First Order Upwind	
Turbulent Kinetic Energy	
First Order Upwind 🔹	
Transient Formulation	
· · · · · · · · · · · · · · · · · · ·	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
High Order Term Relaxation Options	
Default	
Help	

- a. Select Coupled from the Scheme drop-down list.
- b. Confirm that **PRESTO!** is selected from the **Pressure** drop-down list.

The PRESTO! method for pressure is a good choice when buoyancy and inertial forces are present.

2. Set the solution controls.

✤Solution Controls

- a. Enter 40 for Flow Courant Number.
- b. Enter 0.5 for both Momentum and Pressure in the Explicit Relaxation Factors group box.

Solution Controls				
Flow Courant Number				
40				
Explicit	Relaxati	ion Factors		
Mome	ntum 0).5		
Pres	ssure).5		
Under-Relaxation	Factors	5		
Density			4	
1				
Body Forces				
1				
Slip Velocity				
0.1				
Volume Fraction	ı			
0.4				
Turbulent Kinet	ic Energ	у		
0.8				
Default Equations	Limits	Advanced	Ŧ	
Help				

- c. Enter 0.4 for Volume Fraction in the Under-Relaxation Factors group box.
- 3. Enable the plotting of residuals during the calculation.

 $\textcircled{} Monitors \rightarrow \overleftarrow{\sqsubseteq} Residuals \rightarrow Edit...$

Residual Monitors					-X
Options	Equations				
▼ Print to Console	Residual	Monitor (Check Convergence	Absolute Criteria	^
V Plot	continuity	V		1e-05	
Window 1 Curves Axes	x-velocity	V		0.001	
Iterations to Plot	y-velocity	V		0.001	
1000	k			0.001	Ŧ
	Residual Values			Convergence Cr	riterion
Iterations to Store	Normalize		Iterations	absolute	•
	📝 Scale				
	Compute Loca	al Scale			
OK Plot Renormalize Cancel 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.

Solution Initialization

Solution Initialization				
Initialization Methods Hybrid Initialization Standard Initialization 				
More Settings Initialize Patch Reset DPM Sources Reset Statistics				
Help				

- a. Select Hybrid Initialization from the Initialization Methods group box.
- 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.

CRun Calculation

7. Save the case and data files (tee_la.cas.gz and tee_la.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

=

8. Check the total mass flow rate for each phase.

Image: Reports → Im	Fluxes \rightarrow	Set Up

E Flux Reports		— ×
Flux Reports Options Mass Flow Rate Total Heat Transfer Rate Radiation Heat Transfer Rate Phase water Phase water Boundary Types axis exhaust-fan fan inlet-vent Boundary Name Pattern Save Output Parameter	Boundaries 🖹 🗐 🚍 default-interior outflow-3 outflow-5 velocity-inlet-4 wall-1	Results -14.06627843298836 -23.34924479649739 37.41752521252892
Compute Write	Close	q

- 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.

22.4.10. Postprocessing for the Mixture Solution

1. Display the static pressure field in the tee (Figure 22.3: Contours of Static Pressure (p. 941)).

Graphics and Anii	mations $\rightarrow \stackrel{\frown}{=}$ Contours \rightarrow Set Up	
Contours	×	
Options	Contours of Pressure	
 ✓ Node Values ✓ Global Range ✓ Auto Range 	Static Pressure Phase	
Clip to Range	mixture	
Draw Mesh	0	
Levels Setup 20 1 Surface Name Pattern Match	Surfaces	
	New Surface	
	Surface Types	
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**.

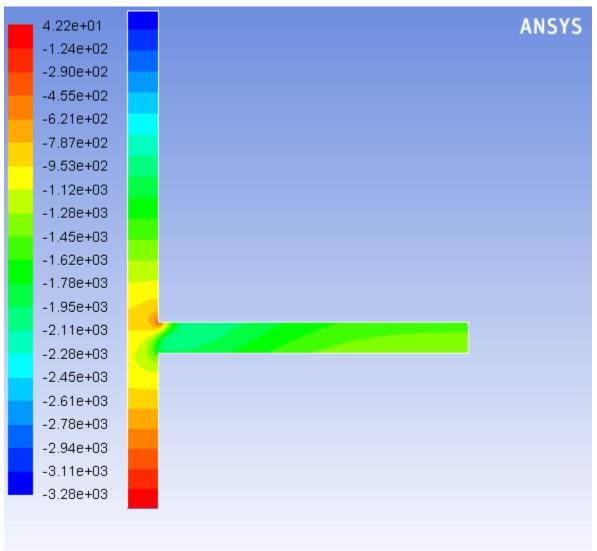


Figure 22.3: Contours of Static Pressure

Contours of Static Pressure (mixture) (pascal)

ANSYS Fluent (2d, dp, pbns, mixture, rke)

In Figure 22.3: Contours of Static Pressure (p. 941) 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 22.4: Contours of Velocity Magnitude (p. 942)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

- 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**.

	1.80e+00	A
ł	1.71e+00	
	1.62e+00	
	1.53e+00	
	1.44e+00	
	1.35e+00	
	1.26e+00	
	1.17e+00	
	1.08e+00	
	9.93e-01	
	9.02e-01	
	8.12e-01	
	7.22e-01	
	6.32e-01	
	5.41e-01	
	4.51e-01	
	3.61e-01	
	2.71e-01	
	1.80e-01	
	9.02e-02	
	0.00e+00	

Figure 22.4: Contours of Velocity Magnitude

Contours of Velocity Magnitude (water) (m/s)

ANSYS Fluent (2d, dp, pbns, mixture, rke)

3. Display the volume fraction of air (Figure 22.5: Contours of Air Volume Fraction (p. 943)).

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

- 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.

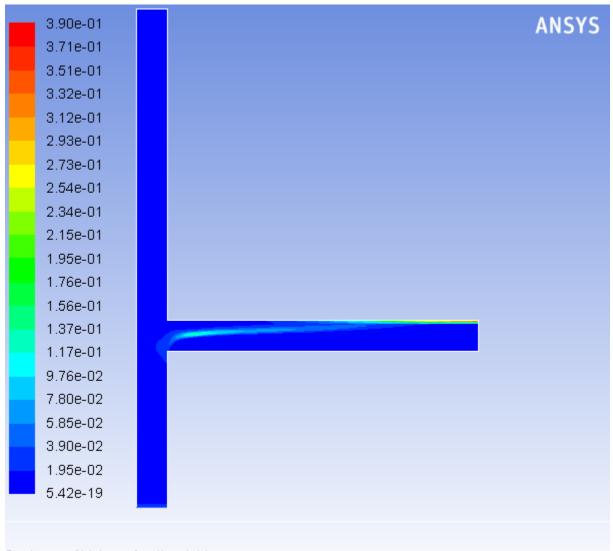


Figure 22.5: Contours of Air Volume Fraction

Contours of Volume fraction (air)

ANSYS Fluent (2d, dp, pbns, mixture, rke)

When gravity acts downwards, it induces stratification in the side arm of the tee junction. In Figure 22.5: Contours of Air Volume Fraction (p. 943), 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.

22.4.11. 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

Using the Mixture and Eulerian Multiphase Models

	Setting	Value
Spatial Discretiza-	Pressure	PRESTO!
tion	Momentum	Third-Order MUSCL
	Volume Fraction	QUICK
	Turbulent Kinetic Energy	Third-Order MUSCL
	Turbulent Dissipation Rate	Third-Order MUSCL

- 2. Run the calculation for an additional 1400 iterations.
- 3. Save the case and data files as tee_lb.cas.gz and tee_lb.dat.gz
- 4. Plot the contours of air volume fraction using the higher order method on the same scale as in Figure 22.5: Contours of Air Volume Fraction (p. 943).

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

- 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 3.90e-1 for Min and Max, respectively.
- e. Click Display and close the Contours dialog box.

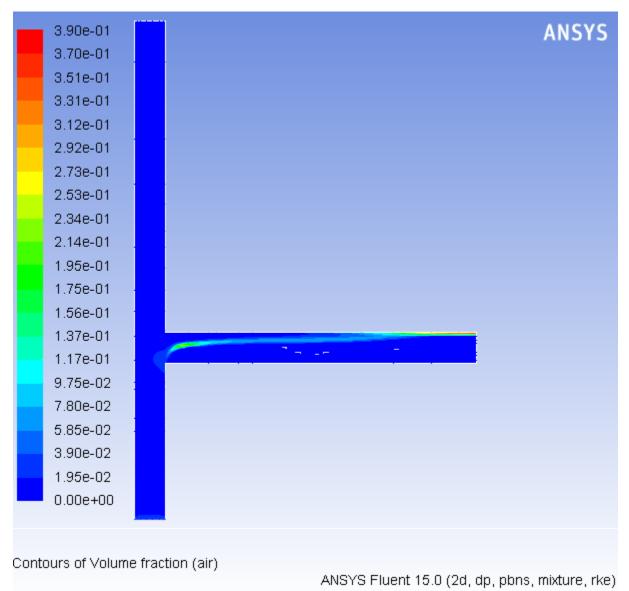


Figure 22.6: Contours of Air Volume Fraction — Higher Order Solution

22.4.12. 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.

 $\mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Multiphase} \to \mathsf{Edit...}$

Multiphase Model	×
Model Off Volume of Fluid Mixture Eulerian Wet Steam Eulerian Parameters	Number of Eulerian Phases
Dense Discrete Phase Model Boiling Model Multi-Fluid VOF Model	
Volume Fraction Parameters Scheme © Explicit @ Implicit	
OK	I 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.

\mathbf{P} Phases \rightarrow Interaction...

Phase Interaction	on		
Virtual Mass			
Drag Lift V	Wall Lubrication Turbulent D	Dispersion Collisions Slip Heat Mass F	Reactions Surface Tension Discretization Interfacial Area
Drag Coefficient			
Drag Modifica	ation		^
air	water	schiller-naumann	▼ Edit
1			
		OK Cancel Help	

- a. Retain the default selection of schiller-naumann from the Drag Coefficient drop-down list.
- 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 Steps for Using a Multiphase Model in the User's Guide.

3. Modify the boundary conditions at the velocity inlet (velocity-inlet-4) for the mixture.

◆Boundary Conditions → Evelocity-inlet-4

- a. Select mixture from the Phase drop-down list.
- b. Click Edit... to open the Velocity Inlet dialog box.
- c. Enter 10 % for **Turbulent Intensity** and click **OK** to close the **Velocity Inlet** dialog box.
- 4. Select the multiphase turbulence model.

Models →
 Eviscous → Edit...

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.
- 5. 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:

	Setting	Value
Pressure-Velocity Coupling	Scheme	Coupled
Spatial Discretiza- tion	Momentum	Third-Order MUSCL
	Volume Fraction	QUICK
	Turbulent Kinetic Energy	Third-Order MUSCL
	Turbulent Dissipation Rate	Third-Order MUSCL

6. Set the solution controls

✤Solution Controls

Solution Controls	
Flow Courant Number	
40	
Explicit Relaxation Factors	
Momentum 0.5	
Pressure 0.5	
Under-Relaxation Factors	
Density	
1	
Body Forces	
1	
Volume Fraction	=
0.4	
Turbulent Kinetic Energy	
0.8	
Turbulent Dissipation Rate	Ĩ
0.8	
	-
Default	
Equations Limits Advanced	
Help	

- 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.
- 7. Continue the solution by requesting 1400 additional iterations.

CRun Calculation

8. Check that the mass imbalance is small (less than about 0.2 %) using the **Flux Reports** dialog box as for the Mixture model solution.

 $\textcircled{P} Reports \rightarrow \overleftarrow{E} Fluxes \rightarrow Set Up...$

9. Save the case and data files (tee_2.cas.gz and tee_2.dat.gz).

File \rightarrow Write \rightarrow Case & Data...

22.4.13. Postprocessing for the Eulerian Model

\bigcirc Graphics and Animations → **\sqsubseteq** Contours → Set Up...

1. Display the static pressure field in the tee for the mixture (Figure 22.7: Contours of Static Pressure — Eulerian Model (p. 951)).

Contours		X
Options	Contours of	
V Filled	Pressure	•
Node Values	Static Pressure	
Global Range	Phase	
Clip to Range	mixture 👻	
Draw Profiles	Min (pascal)	Max (pascal)
Drawmean	-3.28e3	4.22e1
Levels Setup 20 1 Surface Name Pattern Match	Surfaces default-interior outflow-3 outflow-5 velocity-inlet-4 wall-1	
Matur	New Surface	
	Surface Types	
	axis clip-surf	
	exhaust-fan	
	fan	-
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 22.3: Contours of Static Pressure (p. 941).
- d. Click Display.

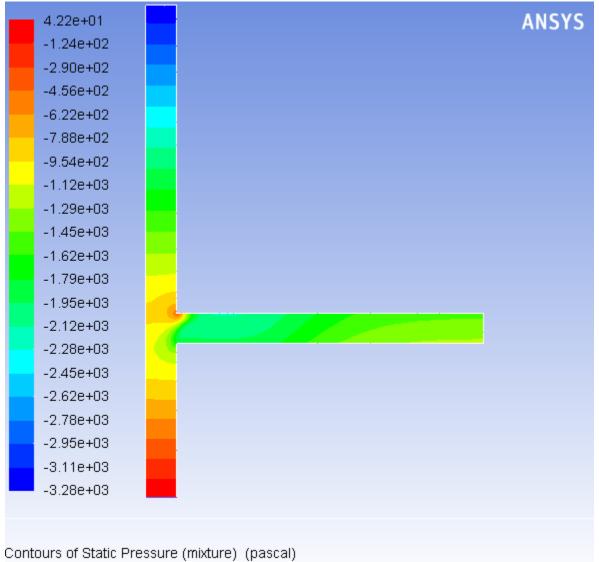


Figure 22.7: Contours of Static Pressure — Eulerian Model

ANSYS Fluent (2d, dp, pbns, eulerian, rke)

2. Display contours of velocity magnitude for water (Figure 22.8: Contours of Water Velocity Magnitude — Eulerian Model (p. 952)).

Graphics and Animations → \blacksquare Contours → Set Up...

- 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 scale to match that in Figure 22.4: Contours of Velocity Magnitude (p. 942).
- d. Click Display.

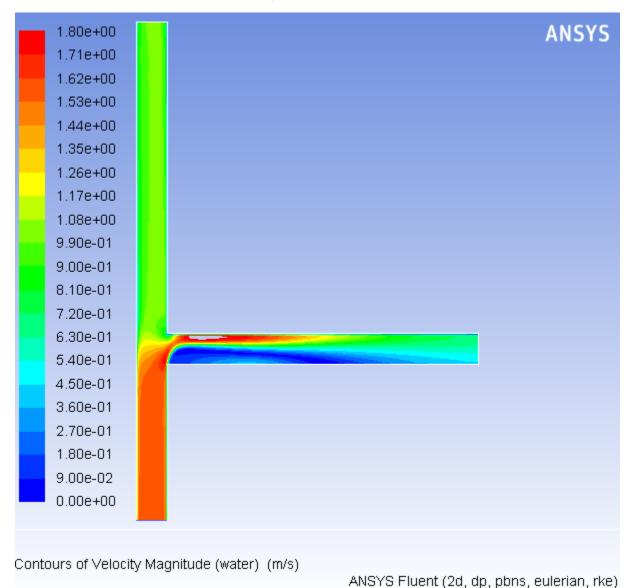


Figure 22.8: Contours of Water Velocity Magnitude — Eulerian Model

3. Display the volume fraction of air (Figure 22.9: Contours of Air Volume Fraction — Eulerian model (p. 953)).

 \clubsuit Graphics and Animations $\rightarrow \Xi$ Contours \rightarrow Set Up...

- 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 scale to match that in Figure 22.5: Contours of Air Volume Fraction (p. 943).
- d. Click **Display** and close the **Contours** dialog box.

3.90e-01	AN
3.70e-01	
3.51e-01	
3.31e-01	
3.12e-01	
2.92e-01	
2.73e-01	
2.53e-01	
2.34e-01	
2.14e-01	
1.95e-01	
1.75e-01	
1.56e-01	
1.37e-01	
1.17e-01	
9.75e-02	
7.80e-02	
5.85e-02	
3.90e-02	
1.95e-02	
0.00e+00	

Figure 22.9: Contours of Air Volume Fraction — Eulerian model

Contours of Volume fraction (air)

Compare the volume fraction plot in Figure 22.9: Contours of Air Volume Fraction — Eulerian model (p. 953) with the volume fraction plot using the mixture model in Figure 22.6: Contours of Air Volume Fraction — Higher Order Solution (p. 945). 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.

22.5. 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

ANSYS Fluent (2d, dp, pbns, eulerian, rke)

comparison of the two approaches. For more information about the mixture and Eulerian models, see Modeling Multiphase Flows in the User's Guide.

22.6. 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. 123).

Chapter 23: Using the Eulerian Multiphase Model for Granular Flow

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
- 23.6. Further Improvements

23.1. Introduction

Mixing tanks are used to maintain solid particles or droplets of heavy fluids in suspension. Mixing may be required to enhance reaction during chemical processing or to prevent sedimentation. In this tutorial, you will use the Eulerian multiphase model to solve the particle suspension problem. The Eulerian multiphase model solves momentum equations for each of the phases, which are allowed to mix in any proportion.

This tutorial demonstrates how to do the following:

- Use the granular Eulerian multiphase model.
- Specify fixed velocities with a user-defined function (UDF) to simulate an impeller.
- Set boundary conditions for internal flow.
- Calculate a solution using the pressure-based solver.
- Solve a time-accurate transient problem.

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:

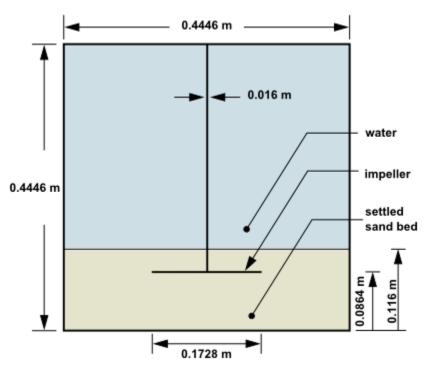
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

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

23.3. Problem Description

The problem involves the transient startup of an impeller-driven mixing tank. The primary phase is water, while the secondary phase consists of sand particles with a 111 micron diameter. The sand is initially settled at the bottom of the tank, to a level just above the impeller. A schematic of the mixing tank and the initial sand position is shown in Figure 23.1: Problem Specification (p. 956). The domain is modeled as 2D axisymmetric.





The fixed-values option will be used to simulate the impeller. Experimental data are used to represent the time-averaged velocity and turbulence values at the impeller location. This approach avoids the need to model the impeller itself. These experimental data are provided in a user-defined function.

23.4. Setup and Solution

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

23.4.1. Preparation 23.4.2. Mesh 23.4.3. General Settings 23.4.4. Models 23.4.5. Materials 23.4.6. Phases 23.4.7. User-Defined Function (UDF) 23.4.8. Cell Zone Conditions 23.4.9. Solution 23.4.10. Postprocessing

23.4.1. 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 15.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip eulerian_multiphase_granular_R150.zip to your working folder.

The files, mixtank.msh and fix.c can be found in the eulerian_multiphase_granular 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 Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Enable **Double-Precision**.
- 11. Ensure Serial is selected under Processing Options.

Note

The double precision solver is recommended for modeling multiphase flow simulations.

23.4.2. Mesh

1. Read the mesh file mixtank.msh.

$\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

A warning message will be displayed twice in the console. You need not take any action at this point, as the issue will be rectified when you define the solver settings in General Settings (p. 958).

23.4.3. General Settings

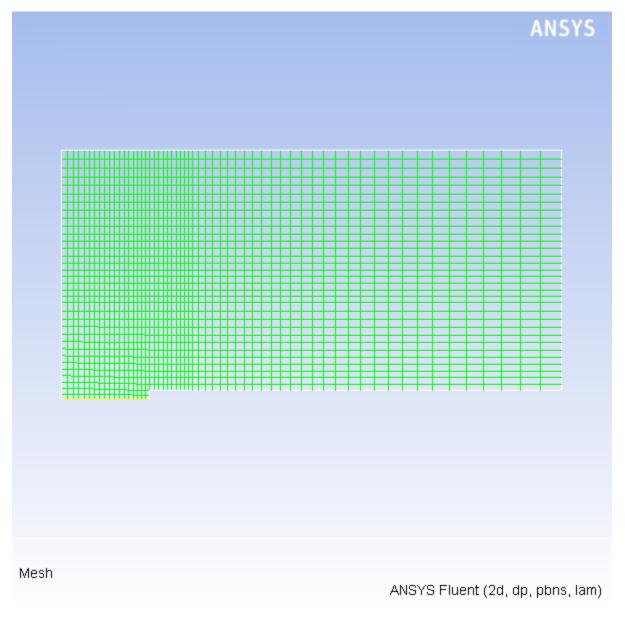
1. Check the mesh.

$\textcircled{} General \rightarrow Check$

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.

2. Examine the mesh (Figure 23.2: Mesh Display (p. 958)).

Figure 23.2: Mesh Display



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 console. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.

3. Modify the mesh colors.

General → Display...

a. Click the Colors... button to open the Mesh Colors dialog box.

You can control the colors used to draw meshes by using the options available in the **Mesh Colors** dialog box.

💶 Mesh Colors		×
Options Color by Type Color by ID Sample	Types far-field inlet interior outlet periodic rans-les-interface	Colors
	symmetry axis wall free-surface internal traction	ved ved ved tan white yellow ▼
Reset Colors	Close	Help

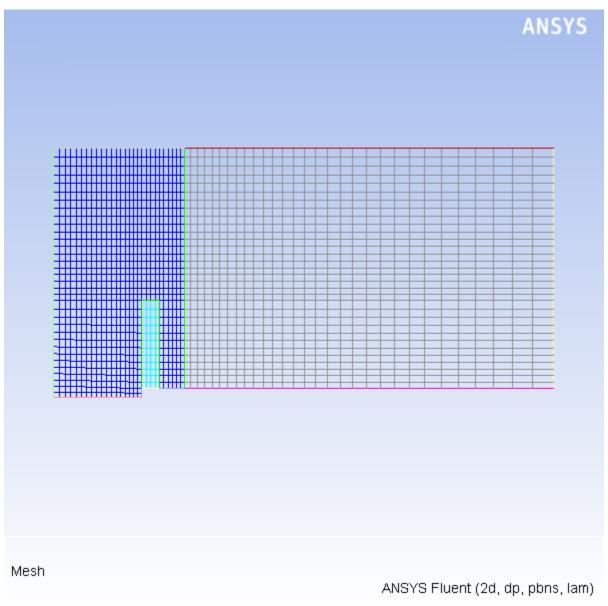
i. Select Color by ID in the Options list.

This will assign a different color to each zone in the domain, rather than to each type of zone.

- ii. Close the Mesh Colors dialog box.
- b. Click **Display** and close the **Mesh Display** dialog box.

The graphics display will be updated to show the mesh.





4. Modify the view of the mesh display to show the full tank upright.

Graphics and Animations → Views...

Q Views		
Views back front Save Name view-0	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes 🖹 🗐 🗐
Apply Came	ra Close	Help

a. Select axis from the Mirror Planes selection list and click Apply.

The mesh display will be updated to show both sides of the tank.

b. Click Auto Scale.

This option is used to scale and center the current display without changing its orientation (Figure 23.4: Mesh Display of the Tank, Mirrored and Scaled (p. 962)).

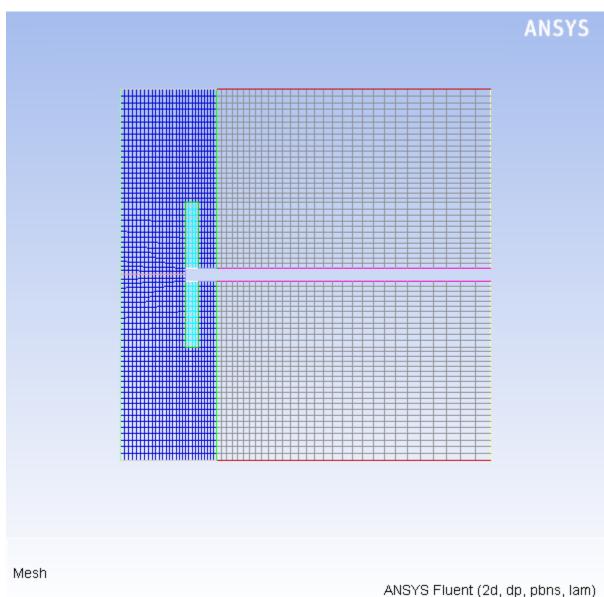


Figure 23.4: Mesh Display of the Tank, Mirrored and Scaled

c. Click the **Options...** button to open the **Display Options** dialog box and select **All** from the **Animation Option** drop-down list. Click **Apply** and close the **Display Options** dialog box.

This will ensure that the 2D geometry remains visible while you manipulate the camera view in the next step.

d. Click the Camera... button to open the Camera Parameters dialog box.

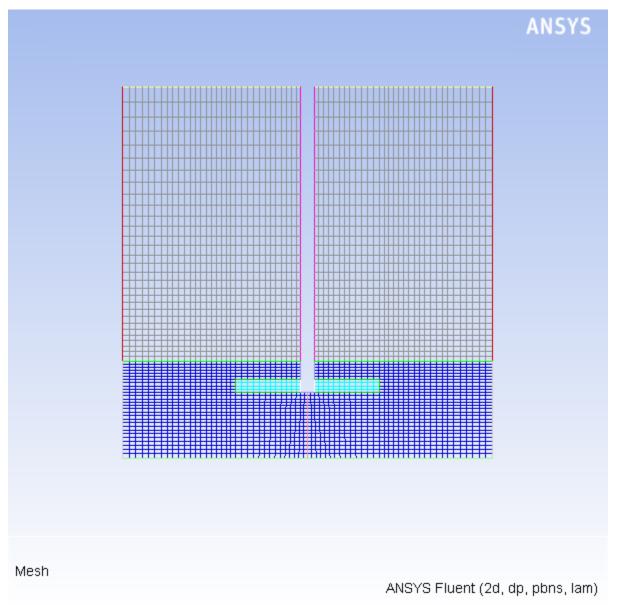
Camera Parameters		×
Camera Position	Projection perspective	•
X (m) 0.2223	* <u></u> *	^
Y (m) 0		
Z (m) 1.571898		ب ا
Apply	Close Help	

- i. Drag the indicator of the dial with the left mouse button in the counter-clockwise direction until the upright view is displayed (Figure 23.5: Mesh Display of the Upright Tank (p. 964)).
- ii. Click Apply and close the Camera Parameters dialog box.
- e. Close the Views dialog box.

Note

While modifying the view, you may accidentally lose the view of the geometry in the display. You can easily revert to the default (front) view by clicking the **Default** button in the **Views** dialog box.





5. Specify a transient, axisymmetric model.

⇔General

General				
Mesh				
Scale	·]	Check	Repor	rt Quality
Displa	iy			
Solver				
Type Pressue Densit	ure-Based ty-Based	Velocity For Absolute Relative	e	n
Time 2D Space Steady Planar Transient Axisymmetric Axisymmetric Swirl				
Gravity				Units
Gravitation	al Accelera	ation		
X (m/s2)	-9.81		P	
Y (m/s2)	0		P	
Z (m/s2)	0		P	
Help				

a. Retain the default Pressure-Based solver.

The pressure-based solver must be used for multiphase calculations.

- b. Select Transient in the Time list.
- c. Select Axisymmetric in the 2D Space list.
- 6. Set the gravitational acceleration.
 - a. Enable Gravity.
 - b. Enter -9.81 m/s^2 for the **Gravitational Acceleration** in the **X** direction.

23.4.4. Models

1. Enable the Eulerian multiphase model.

$$\mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Multiphase} \to \mathsf{Edit...}$$

Multiphase Model	×
Model Off Volume of Fluid Mixture Eulerian Wet Steam Eulerian Parameters Dense Discrete Phase Model Boiling Model Multi-Fluid VOF Model Volume Fraction Parameters Scheme	Number of Eulerian Phases 2
 Explicit Implicit 	
OK Cance	l Help

- a. Select Eulerian in the Model list.
- b. Retain the default setting of 2 for Number of Eulerian Phases.
- c. Click **OK** to close the **Multiphase Model** dialog box.
- 2. Enable the k- ε turbulence model with standard wall functions.

♦ Models → Edit...

Viscous Model	
Model Laminar K-epsilon (2 eqn) K-epsilon Stress (5 eqn) K-epsilon Model Standard RNG Realizable Near-Wall Treatment Scalable Wall Functions Scalable Wall Functions Scalable Wall Functions Contemport Wall Treatment User-Defined Wall Functions Options Production Kato-Launder Production Limiter Turbulence Multiphase Model Mixture Mixture Pre Phase	Model Constants
OK	Cancel Help

a. Select k-epsilon (2eqn) in the Model list.

b. Retain Standard Wall Functions in the Near-Wall Treatment list.

This problem does not require a particularly fine mesh hence, standard wall functions can be used.

c. Select Dispersed in the Turbulence Multiphase Model list.

The dispersed turbulence model is applicable in this case because there is clearly one primary continuous phase and the material density ratio of the phases is approximately 2.5. Furthermore, the Stokes number is much less than 1. Therefore, the kinetic energy of the particle will not differ significantly from that of the liquid. For more information, see Model Comparisons in the Theory Guide.

d. Click **OK** to close the **Viscous Model** dialog box.

23.4.5. Materials

In this step, you will add liquid water to the list of fluid materials by copying it from the ANSYS Fluent materials database and create a new material called sand.

1. Copy liquid water from the Fluent materials database so that it can be used for the primary phase.

```
 \diamondsuit Materials \rightarrow \blacksquare Fluid \rightarrow Create/Edit...
```

a. Click the Fluent Database... button to open the Fluent Database Materials dialog box.

Fluent Database Materials		x
Fluent Fluid Materials vinyl-silylidene (h2cchsih) vinyl-trichlorosilane (sid3ch2ch) vinylidene-chloride (ch2cd2) water-liquid (h2o<1>) water-vapor (h2o) wood-volatiles (wood_vol) (Material Type fluid Order Materials by Name Chemical Formula 	
Properties Density (kg/m3) Cp (Specific Heat) (j/kg-k)	998.2	-
Thermal Conductivity (w/m-k) Viscosity (kg/m-s)	constant View 0.6 View	
New Edit	constant View 0.001003 Save Copy Close Help	Ŧ

b. 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>).

- c. Click **Copy** to copy the information for liquid water to your model.
- d. Close the Fluent Database Materials dialog box.
- 2. Create a new material called **sand**.

Create/Edit Mat	erials			—
Name		Material Type		Order Materials by
sand		fluid		Name
Chemical Formula		Fluent Fluid Materials		Chemical Formula
		water-liquid (h2o <l>)</l>		Fluent Database
		Mixture		User-Defined Database
		none		~
Properties				
Density (kg/m3)	constant	▼ Edit	Â	
	2500			
Viscosity (kg/m-s)	constant	Edit		
	0.001003			
	0.001003		=	
			-	
1			•	
	Change/Create	Delete Close	e Help	

- a. Enter sand for Name and delete the entry in the Chemical Formula field.
- b. Enter 2500 kg/m³ for **Density** in the **Properties** group box.
- c. Click Change/Create.

A **Question** dialog box will open, asking if you want to overwrite **water-liquid**.

d. Click **No** in the **Question** dialog box to retain **water-liquid** and add the new material (**sand**) to the list.

The **Create/Edit Materials** dialog box will be updated to show the new material, **sand**, in the **Fluent Fluid Materials** drop-down list.

3. Close the Create/Edit Materials dialog box.

23.4.6. Phases

Phases

Phases

Phases	
phase-1 - Primary Phase	
phase-2 - Secondary Phase	
Edit Interaction	ID 2
[]	
Help	

1. Specify water (water-liquid) as the primary phase.

$\mathbf{O} \mathsf{Phases} \to \mathbf{F} \mathsf{phase-1} \to \mathsf{Edit...}$

Primary Phase	×
Name	
water	
Phase Material water-liquid	
OK 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 sand (sand) as the secondary phase.

 $\diamondsuit Phases \rightarrow \blacksquare phase-2 \rightarrow Edit...$

Secondary Phase			×
Name		_	
sand			
Phase Material sand	▼ Edit		
🔽 Granular			
Packed Bed			
Granular Temperature Model	1		
 Phase Property Partial Differential Equation 			
Properties	_		
Solids Pressure (pascal)	lun-et-al 🔹	Edit	*
Radial Distribution	lun-et-al 🔹	Edit	
Elasticity Modulus (pascal)	derived 🔹	Edit	
Packing Limit	constant 🔹	Edit	=
	0.6		
]			Ŧ
ОК	Cancel Help		

- a. Enter sand for Name.
- b. Select sand from the Phase Material drop-down list.
- c. Enable Granular.
- d. Retain the selection of Phase Property in the Granular Temperature Model list.
- e. Enter 0.000111 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. Enter 0.6 for **Packing Limit**.

Scroll down in the **Properties** group box to locate **Packing Limit**.

- i. Click **OK** to close the **Secondary Phase** dialog box.
- 3. Specify the interaction terms between the phases.

 $\mathbf{\Phi}$ Phases \rightarrow Interaction...

Phase Interaction	1						×
Virtual Mass							
Drag Lift We	I Lubrication Turbulent (Rispersion Turbulence Interaction C	ollisions Slip Heat	Mass Rea	ctions Surface Tensio	on Discretization	Interfacial Area
Drag Coefficient				-			
Drag Modificati	on			^			
sand	water	gidaspow	• Edit				
		1		*			
OK Cancel Help							

- a. In the Drag tab, select gidaspow from the Drag Coefficient drop-down list.
- b. In the **Turbulence Interaction** tab, select **simonin-et-al** from the **Turbulence Interaction** drop-down list.

The Simonin-et-al Model dialog box will appear. Click OK to retain the default model settings.

c. Click OK to close the Phase Interaction dialog box.

23.4.7. User-Defined Function (UDF)

A UDF is used to specify the fixed velocities that simulate the impeller. The values of the time-averaged impeller velocity components and turbulence quantities are based on experimental measurement. The variation of these values may be expressed as a function of radius, and imposed as polynomials according to:

$$variable = A_1 + A_2 r + A_3 r^2 + A_4 r^3 + \dots$$

The order of polynomial to be used depends on the behavior of the function being fitted. For this tutorial, the polynomial coefficients shown in Table 23.1: Impeller Profile Specifications (p. 972)

Variable	A1	A2	A3	A4	A5	A6
u velocity	-7.1357e-2	54.304	-3.1345e+3	4.5578e+4	-1.966e+5	-
v velocity	3.1131e-2	-10.313	9.5558e+2	-2.0051e+4	1.186e+5	-
kinetic energy	2.2723e-2	6.7989	-424.18	9.4615e+3	-7.725e+4	1.8410e+5
dissipation	-6.5819e-2	88.845	-5.3731e+3	1.1643e+5	-9.120e+5	1.9567e+6

Table 23.1: Impeller Profile Specifications

For more information about setting up a UDF using the DEFINE_PROFILE macro, refer to the separate UDF Manual. Though this macro is usually used to specify a profile condition on a boundary face zone, it is used in fix.c to specify the condition in a fluid cell zone. Hence, the arguments of the macro have been changed accordingly.

1. Interpret the UDF source file fix.c.

Define \rightarrow **User-Defined** \rightarrow **Functions** \rightarrow **Interpreted...**

Interpreted UDFs	-
Source File Name	
fix.c	Browse
CPP Command Name	
срр	
Stack Size	
 Display Assembly Listing Use Contributed CPP 	
Interpret Close	Help

a. Enter fix.c for Source File Name.

If the UDF source file is not in your working folder, you must enter the entire folder path for **Source File Name** instead of just entering the file name. Alternatively, click **Browse...** and select **fix.c** in the eulerian_multiphase_granular folder that was created after you unzipped the original file.

b. Enable Display Assembly Listing.

The **Display Assembly Listing** option displays the assembly language code in the console as the function compiles.

- c. Click Interpret to interpret the UDF.
- d. Close the Interpreted UDFs dialog box.

Note

The name and contents of the UDF are stored in the case file when you save the case file.

23.4.8. Cell Zone Conditions

Cell Zone Conditions

Cell Zone Conditions				
Zone				
fix-zone fluid initial-sand				
Phase	Туре	ID		
water	✓ fluid ✓	4		
Edit Parameters Display Mesh Porous Formulation Superficial Velocit Physical Velocit	city			
Help				

For this problem, you do not have to specify any conditions for outer boundaries. Within the domain, there are three fluid zones, representing the impeller region, the region where the sand is initially located, and the rest of the tank. There are no conditions to be specified in the latter two zones, so you need to set conditions only in the zone representing the impeller.

1. Set the boundary conditions for the fluid zone representing the impeller (**fix-zone**) for the primary phase.

Cell Zone Conditions → 🔄 fix-zone

You will specify the conditions for water and sand separately using the UDF. The default conditions for the mixture (that is, the conditions that apply to all phases) are acceptable.

- a. Select water from the Phase drop-down list.
- b. Click the **Edit...** button to open the **Fluid** dialog box.

Inid			
Zone Name	Phase		
fix-zone	water		
Porous Zone Source Terms			
Fixed Values			
Reference Frame Mesh Motion Porous Zone Embedded LES	Reaction Source Terms Fixed Values Multiphase		
Axial Velocity (m/s)	udf fixed_u		
Radial Velocity (m/s)	udf fixed_v		
Turbulent Kinetic Energy (m2/s2)	udf fixed_ke		
Turbulent Dissipation Rate (m2/s3)	udf fixed_diss		
OK Cancel Help			

i. Enable Fixed Values.

The **Fluid** dialog box will expand to show the related inputs.

ii. Click the **Fixed Values** tab and set the following fixed values:

Parameter	Value
Axial Velocity	udf fixed_u
Radial Velocity	udf fixed_v
Turbulence Kinetic Energy	udf fixed_ke
Turbulence Dissipation Rate	udf fixed_diss

- c. Click **OK** to close the **Fluid** dialog box.
- 2. Set the boundary conditions for the fluid zone representing the impeller (**fix-zone**) for the secondary phase.

♦ Cell Zone Conditions $\rightarrow \stackrel{\frown}{=} fix$ -zone

- a. Select **sand** from the **Phase** drop-down list.
- b. Click the **Edit...** button to open the **Fluid** dialog box.

💶 Fluid
Zone Name Phase
fix-zone sand
Porous Zone Source Terms
Fixed Values
Reference Frame Mesh Motion Porous Zone Embedded LES Reaction Source Terms Fixed Values Multiphase
Axial Velocity (m/s)
Radial Velocity (m/s) udf fixed_v
OK Cancel Help

i. Enable Fixed Values.

The **Fluid** dialog box will expand to show the related inputs.

ii. Click the Fixed Values tab and set the following fixed values:

Parameter	Value
Axial Velocity	udf fixed_u
Radial Velocity	udf fixed_v

c. Click **OK** to close the **Fluid** dialog box.

23.4.9. Solution

1. Set the under-relaxation factors.

Solution Controls

a. Enter 0.5 for Pressure, 0.2 for Momentum, and 0.8 for Turbulent Viscosity.

Solution Controls	
Under-Relaxation Factors	
Volume Fraction	٠
0.5	
Granular Temperature	
0.2	
Turbulent Kinetic Energy	-
0.8	
Turbulent Dissipation Rate	
0.8	=
Turbulent Viscosity	
0.8	
	Ŧ
Default	
Equations Limits Advanced	
Help	

Tip

Scroll down in the **Under-Relaxation Factors** group box to locate **Turbulent Viscosity**.

2. Enable the plotting of residuals during the calculation.

 $\textcircled{} Monitors \rightarrow \overleftarrow{\blacksquare} Residuals \rightarrow Edit...$

Residual Monitors					X
Options Print to Console Plot Window Curves Iterations to Plot 1000	Equations Residual continuity u-water u-sand v-water	Monitor (Check Convergence	Absolute Criteria 0.001 0.001 0.001 0.001 0.001	• III
Iterations to Store	Residual Values Normalize Scale Compute Loca	al Scale	Iterations 5	Convergence Cr absolute	iterion
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.
- 3. Initialize the solution using the default initial values.

Solution Initialization

Solution Initialization	
Initialization Methods	
Hybrid Initialization	
 Standard Initialization 	
Compute from	
	-
Reference Frame	
 Relative to Cell Zone Absolute 	
Initial Values	
Gauge Pressure (pascal)	4
0	
water Axial Velocity (m/s)	
	Ξ
water Radial Velocity (m/s)	
0	
water Turbulent Kinetic Energy (m2/s2)	
1	
water Turbulent Dissipation Rate (m2/s3)	
1	
sand Axial Velocity (m/s)	
0	
	Ŧ
Initialize Reset Patch	
Reset DPM Sources Reset Statistics	
Help	

- a. Retain the default initial values and click Initialize.
- 4. Patch the initial sand bed configuration.

Solution Initialization → Patch...

Patch		X
Reference Frame Reference Frame Reference Frame Absolute Phase Sand Variable Axial Velocity Radial Velocity Granular Temperature Volume Fraction	Value 0.3 Use Field Function Field Function	Zones to Patch
Patc	h Close Help	

- a. Select sand from the Phase drop-down list.
- b. Select Volume Fraction from the Variable selection list.
- c. Enter 0.3 for Value.
- d. Select initial-sand from the Zones to Patch selection list.
- e. Click Patch and close the Patch dialog box.
- 5. Save the initial case and data files (mixtank.cas.gz and mixtank.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

The problem statement is now complete. As a precaution, you should review the impeller velocity fixes and sand bed patch after running the calculation for a single time step. Since you are using a UDF for the velocity profiles, perform one time step in order for the profiles to be calculated and available for viewing.

6. Set the time stepping parameters and run the calculation for 0.005 seconds.

Run Calculation

- a. Enter 0.005 for Time Step Size.
- b. Enter 1 for Number of Time Steps.

Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method Fixed •	Time Step Size (s)
Settings	Number of Time Steps
Options	
Extrapolate Variables Data Sampling for Time Sampling Interval	Sampling Options
Max Iterations/Time Step 40	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- c. Enter 40 for Max Iterations/Time Step.
- d. Click Calculate.
- 7. Examine the initial velocities and sand volume fraction.

In order to display the initial fixed velocities in the fluid zone (**fix-zone**), you need to create a surface for this zone.

a. Create a surface for **fix-zone**.

Surface → Zone...

Zone Surface	
Zone axis axis default-interior default-interior:014 default-interior:015 default-interior:016 default-interior:017 fix-zone fluid initial-sand wall-1 wall-1:013 wall-2 	Surfaces axis default-interior default-interior:014 default-interior:015 default-interior:016 default-interior:017 fix-zone wall
Create Manage	Close Help

i. Select fix-zone from the Zone selection list and click Create.

The default name is the same as the zone name. ANSYS Fluent automatically assign the default name to the new surface when it is created. The new surface is added to the **Surfaces** selection list in the **Zone Surface** dialog box.

- ii. Close the **Zone Surface** dialog box.
- b. Display the initial impeller velocities for water (Figure 23.6: Initial Impeller Velocities for Water (p. 984)).

\bigcirc Graphics and Animations → **\stackrel{\frown}{=}** Vectors → Set Up...

Vectors Options Vectors of Global Range Velocity Auto Range Of to Range Auto Scale Velocity	•
Image Velocity Image Phase Image Image Image	•
Auto Range Phase Clip to Range water Auto Scale Water	•
✓ Auto Range Phase ✓ Clip to Range water ✓ Auto Scale Water	
V Auto Scale	•
Draw Mesh Color by	
Velocity	•
Style	
arrow 🗸 Velocity Magnitude	•
Scale Skip Phase	
1 0 water	-
Min (m/s) Max (m/s)	
Vector Options 0 0.800734	
Custom Vectors Surfaces	
default-interior:016	
Surface Name Pattern default-interior:017	
Match fix-zone	
wall	=
wall-1 wall-1:013	
Wall-1.015	Ψ.
New Surface 🕶	
Surface Types	
axis	
dip-surf	
degassing	
exhaust-fan	Ψ.
Display Compute Close Help	

- i. Retain the selection of **Velocity** from the **Vectors of** drop-down list.
- ii. Retain the selection of **water** from the **Phase** drop-down list below the **Vectors of** drop-down list.
- iii. Retain the selection of Velocity... and Velocity Magnitude from the Color by drop-down lists.
- iv. Retain the selection of water from the Phase drop-down list below the Color by drop-down lists.
- v. Select **fix-zone** from the **Surfaces** selection list and click **Display**.

ANSYS Fluent will display the water velocity vectors fixes at the impeller location, as shown in Figure 23.6: Initial Impeller Velocities for Water (p. 984).

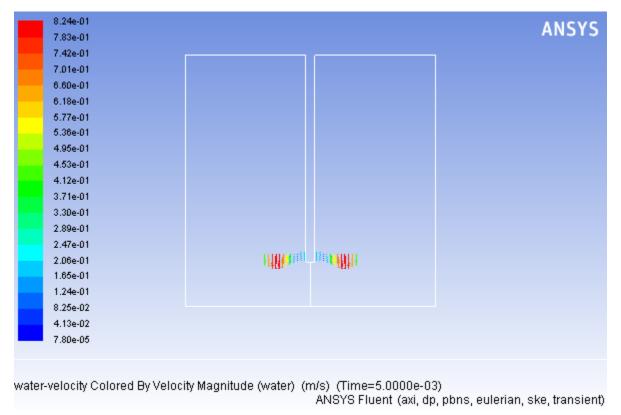


Figure 23.6: Initial Impeller Velocities for Water

c. Display the initial impeller velocities for sand (Figure 23.7: Initial Impeller Velocities for Sand (p. 985)).

Graphics and Animations → $\stackrel{\frown}{=}$ Vectors → Set Up...

- i. Select **sand** from the **Phase** drop-down lists (below the **Vectors of** drop-down list and **Color by** drop-down lists).
- ii. Click **Display** (Figure 23.7: Initial Impeller Velocities for Sand (p. 985)) and close the **Vectors** dialog box.

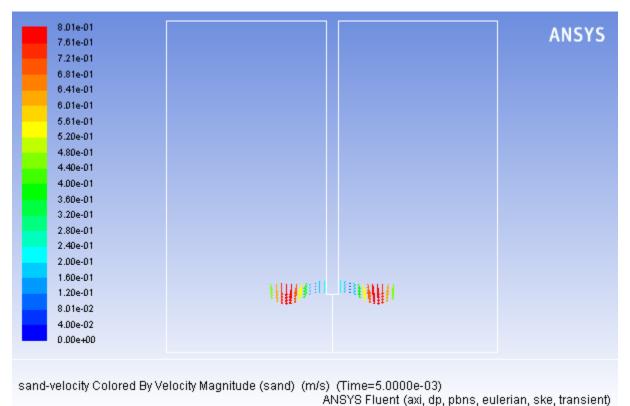


Figure 23.7: Initial Impeller Velocities for Sand

d. Display contours of sand volume fraction (Figure 23.8: Initial Settled Sand Bed (p. 987)).

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

Contours		×	
Options	Contours of		
Clip to Range	Phases	•	
	Volume fraction	•	
	Phase		
	sand	•	
Draw Profiles	Min Max		
Drawmesh	0		
Levels Setup	Surfaces		
20 🛋 1	 ▲ axis ✓ default-interior 	Â	
	default-interior:014		
Surface Name Pattern	default-interior:015 default-interior:016	_	
Mat		•	
	New Surface 💌		
	Surface Types		
	axis	*	
	clip-surf		
	degassing exhaust-fan	-	
	l <u>.</u> .		
Display Compute Close Help			

- i. Enable Filled in the Options group box.
- ii. Select **sand** from the **Phase** drop-down list.
- iii. Select Phases... and Volume fraction from the Contours of drop-down lists.
- iv. Click **Display** and close the **Contours** dialog box.

ANSYS Fluent will display the initial location of the settled sand bed, as shown in Figure 23.8: Initial Settled Sand Bed (p. 987).

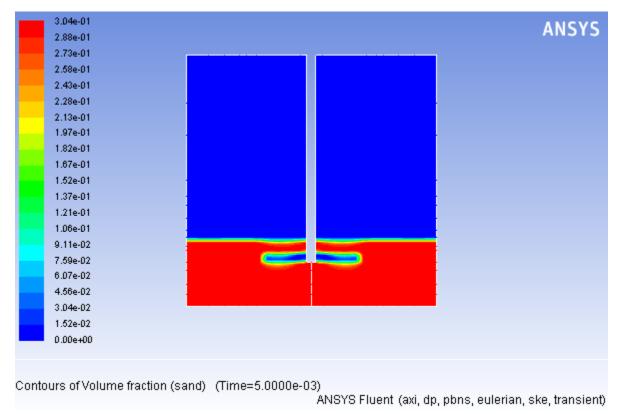


Figure 23.8: Initial Settled Sand Bed

8. Run the calculation for 1 second.

Run Calculation

- a. Enter 199 for Number of Time Steps.
- b. Click Calculate.

After a total of 200 time steps have been computed (1 second of operation), you will review the results before continuing.

9. Save the case and data files (mixtank1.cas.gz and mixtank1.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

- 10. Examine the results of the calculation after 1 second.
 - a. Display the velocity vectors for water in the whole tank (Figure 23.9: Water Velocity Vectors after 1 s (p. 988)).

Graphics and Animations → \blacksquare Vectors → Set Up...

- i. Select **water** from the **Phase** drop-down lists (below the **Vectors of** drop-down list and **Color by** drop-down lists).
- ii. Deselect fix-zone from the Surfaces selection list.

iii. Click Display.

Figure 23.9: Water Velocity Vectors after 1 s (p. 988) shows the water velocity vectors after 1 second of operation. The circulation is confined to the region near the impeller, and has not yet had time to develop in the upper portions of the tank.

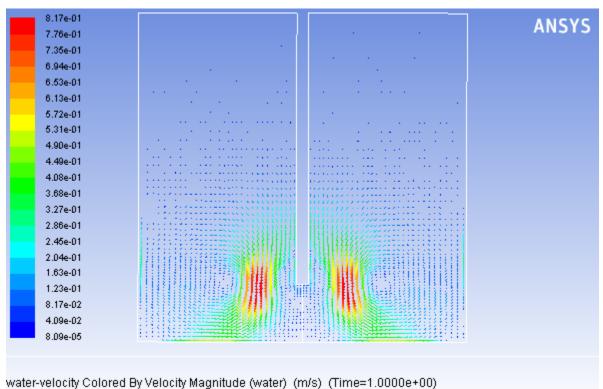


Figure 23.9: Water Velocity Vectors after 1 s

ANSYS Fluent (axi, dp, pbns, eulerian, ske, transient)

b. Display the velocity vectors for the sand (Figure 23.10: Sand Velocity Vectors after 1 s (p. 989)).



- i. Select **sand** from the **Phase** drop-down lists (below the **Vectors of** drop-down list and **Color by** drop-down lists).
- ii. Click **Display** and close the **Vectors** dialog box.

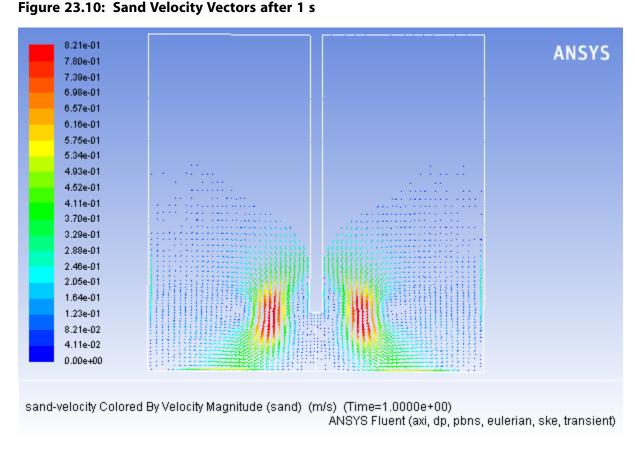


Figure 23.10: Sand Velocity Vectors after 1 s (p. 989) shows the sand velocity vectors after 1 second of operation. The circulation of sand around the impeller is significant, but note that no sand vectors are plotted in the upper part of the tank, where the sand is not yet present.

c. Display contours of sand volume fraction (Figure 23.11: Contours of Sand Volume Fraction after 1 s (p. 990)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

- i. Retain the selection of Phases... and Volume fraction from the Contours of drop-down lists.
- ii. Retain the selection of sand from the Phase drop-down list.
- iii. Click **Display** and close the **Contours** dialog box.

Notice that the action of the impeller draws clear fluid from above the originally settled bed and mixes it into the sand. To compensate, the sand bed is lifted up slightly. The maximum sand volume fraction has decreased as a result of the mixing of water and sand.

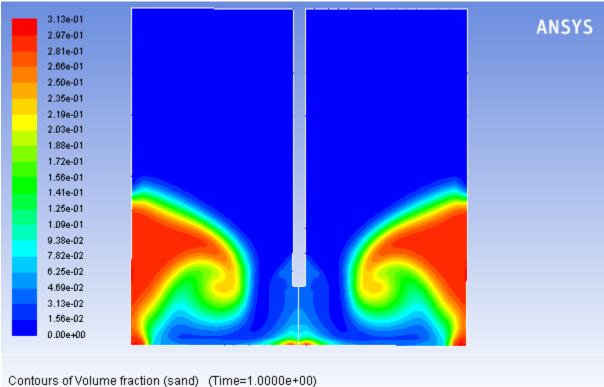


Figure 23.11: Contours of Sand Volume Fraction after 1 s

ANSYS Fluent (axi, dp, pbns, eulerian, ske, transient)

11. Continue the calculation for another 99 seconds.

Run Calculation

a. Set the **Time Step Size** to 0.05.

The initial calculation was performed with a very small time step size to stabilize the solution. After the initial calculation, you can increase the time step to speed up the calculation.

- b. Enter 1980 for Number of Time Steps.
- c. Click Calculate.

The transient calculation will continue up to 100 seconds.

12. Save the case and data files (mixtank100.cas.gz and mixtank100.dat.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}$

23.4.10. Postprocessing

You will now examine the progress of the sand and water in the mixing tank after a total of 100 seconds. The mixing tank has nearly, but not quite, reached a steady flow solution.

1. Display the velocity vectors for water (Figure 23.12: Water Velocity Vectors after 100 s (p. 991)).

\bigcirc Graphics and Animations → **\equiv** Vectors → Set Up...

Figure 23.12: Water Velocity Vectors after 100 s (p. 991) shows the water velocity vectors after 100 seconds of operation. The circulation of water is now very strong in the lower portion of the tank, though modest near the top.

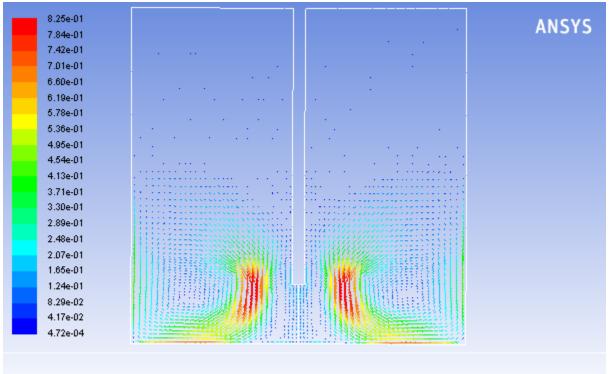


Figure 23.12: Water Velocity Vectors after 100 s

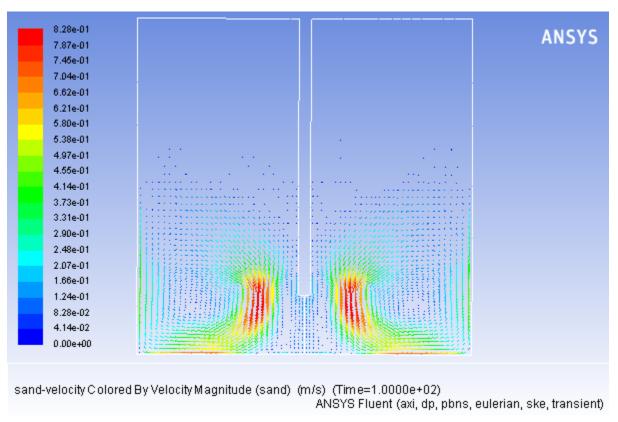
water-velocity Colored By Velocity Magnitude (water) (m/s) (Tim e=1.0000e+02) ANSYS Fluent (axi, dp, pbns, eulerian, ske, transient)

2. Display the velocity vectors for sand (Figure 23.13: Sand Velocity Vectors after 100 s (p. 992)).

Graphics and Animations → \blacksquare Vectors → Set Up...

Figure 23.13: Sand Velocity Vectors after 100 s (p. 992) shows the sand velocity vectors after 100 seconds of operation. The sand has now been suspended much higher within the mixing tank, but does not reach the upper region of the tank. The water velocity in that region is not sufficient to overcome the gravity force on the sand particles.





3. Display contours of sand volume fraction (Figure 23.14: Contours of Sand Volume Fraction after 100 s (p. 993)).



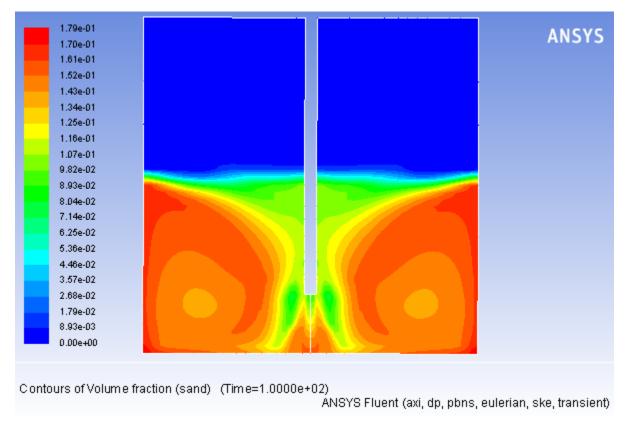


Figure 23.14: Contours of Sand Volume Fraction after 100 s

4. Display filled contours of static pressure for the mixture (Figure 23.15: Contours of Pressure after 100 s (p. 994)).



- a. Select **mixture** from the **Phase** drop-down list.
- b. Select Pressure... and Static Pressure from the Contours of drop-down lists.
- c. Click **Display** and close the **Contours** dialog box.

Figure 23.15: Contours of Pressure after 100 s (p. 994) shows the pressure distribution after 100 seconds of operation. The pressure field represents the hydrostatic pressure except for some slight deviations due to the flow of the impeller near the bottom of the tank.

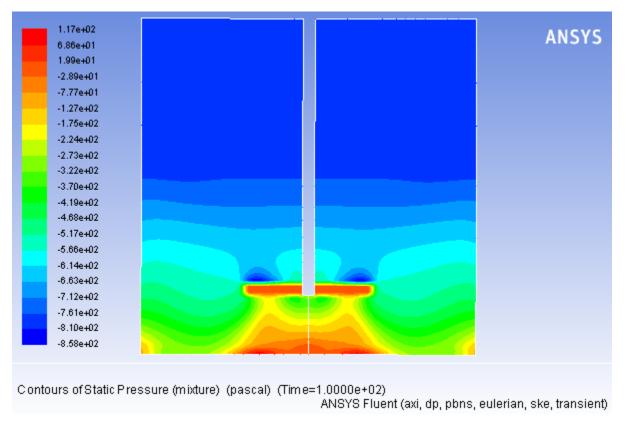


Figure 23.15: Contours of Pressure after 100 s

23.5. Summary

This tutorial demonstrated how to set up and solve a granular multiphase problem using the Eulerian multiphase model. The problem involved the 2D modeling of particle suspension in a mixing tank and postprocessing showed the near-steady-state behavior of the sand in the mixing tank, under the assumptions made.

23.6. 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. 123).

Chapter 24: Modeling Solidification

This tutorial is divided into the following sections:

24.1. Introduction24.2. Prerequisites24.3. Problem Description24.4. Setup and Solution24.5. Summary24.6. Further Improvements

24.1. 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.

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. 123)

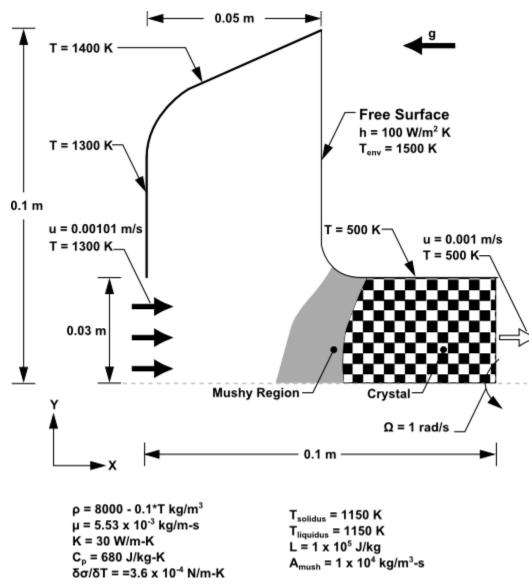
and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

24.3. 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 24.1: Solidification in Czochralski Model (p. 996)), 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 24.1: Solidification in Czochralski Model (p. 996).

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.





 A_{mush} is the mushy zone constant. For details refer to section Momentum Equations for modeling the solidification/melting process, in the Theory Guide.

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 Checking the Mesh
24.4.3. Specifying Solver and Analysis Type

- 24.4.4. Specifying the Models
 24.4.5. Defining Materials
 24.4.6. Setting the Cell Zone Conditions
 24.4.7. Setting the Boundary Conditions
 24.4.8. Solution: Steady Conduction
- 24.4.9. Solution: Transient Flow and Heat Transfer

24.4.1. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip the solidification_R150.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** 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 Fluent Launcher in the Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, 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.

24.4.2. Reading and Checking the Mesh

1. Read the mesh file solid.msh.

$\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$

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 will be 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.

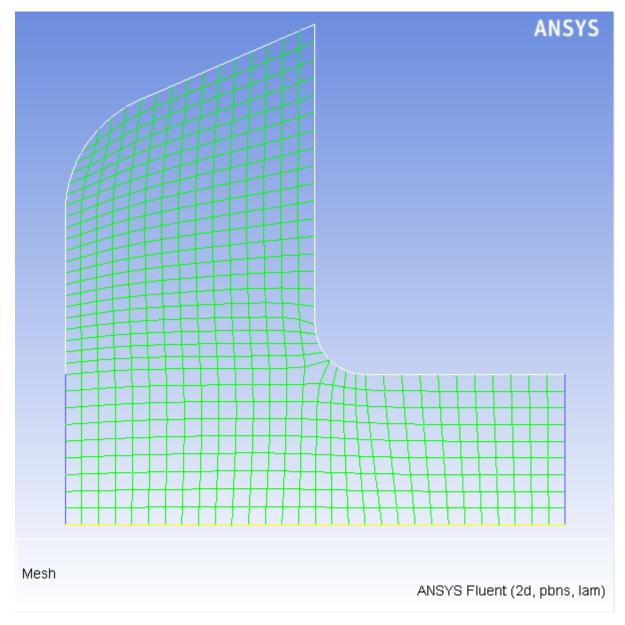
2. Check the mesh.

$\mathbf{O}_{\mathsf{General}} \rightarrow \mathsf{Check}$

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 24.2: Mesh Display (p. 999)).

Figure 24.2: Mesh Display



24.4.3. Specifying Solver and Analysis Type

1. Select **Axisymmetric Swirl** from the **2D Space** list.

General

General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative
Time Steady Transient	2D Space Planar 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.

 $\bigcirc General \rightarrow \mathbf{\overline{M}} Gravity$

General	
Mesh Scale Display	Check Report Quality
Solver	
Type ● Pressure-Based ○ Density-Based	Velocity Formulation
Time	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Gravitational Acceleratio	n
X (m/s2) -9.81	e
Y (m/s2)	e
Z (m/s2)	9
Help	

- a. Enable Gravity.
- b. Enter -9.81 m/s^2 for **X** in the **Gravitational Acceleration** group box.

24.4.4. 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 Velocitie	es	
	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 Setup Procedure in the User's Guide.

d. Click OK to close the Solidification and Melting dialog box.

An **Information** dialog box will open, telling you that available material properties have changed for the solidification model. You will set the material properties later, so you can simply 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.

24.4.5. Defining Materials

In this step, you will create a new material and specify its properties, including the melting heat, solidus temperature, and liquids temperature.

1. Define a new material.



Create/Edit Materials		Order Materials by
liquid-metal	Material Type	Name
	fluid	Chemical Formula
Chemical Formula	Fluent Fluid Materials	
	liquid-metal	 Fluent Database
	Mixture	User-Defined Database
	none	*
Properties		
Density (kg/m3)	polynomial	<u> </u>
Cp (Specific Heat) (j/kg-k)	constant	E
	1006.43	
Thermal Conductivity (w/m-k)	constant Edit	
	0.0242	
Viscosity (kg/m-s)	constant	
	1.7894e-05	
		-

- a. Enter liquid-metal for Name.
- b. Select **polynomial** from the **Density** drop-down list to open the **Polynomial Profile** dialog box.

Scroll down the list to find **polynomial**.

💶 Polynomial Profile				×
Define Density		n Terms of Femperature	Coefficients	
Coefficients				
1 8000	2 -0.1	3	4	
5	6	7	8	
	(e			
	OK	Cancel Help		

- i. Set **Coefficients** to 2.
- ii. Enter 8000 for 1 and -0.1 for 2 in the Coefficients group box.

As shown in Figure 24.1: Solidification in Czochralski Model (p. 996), 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.

A **Question** dialog box will open, asking you if **air** should be overwritten. Click **No** to retain **air** and add the new material (**liquid-metal**) to the **Fluent Fluid Materials** drop-down list.

c. Select **liquid-metal** from the **Fluent Fluid Materials** drop-down list to set the other material properties.

Create/Edit Materials					—
Name		Material Type			Order Materials by
liquid-metal		fluid		•	Name
Chemical Formula		Fluent Fluid Materials			Ochemical Formula
		liquid-metal		•	Fluent Database
		Mixture			User-Defined Database
		none		Ŧ	
Properties				1.	
Viscosity (kg/m-s)	constant	•	Edit	ŕ	
	0.00553				
Pure Solvent Melting Heat (j/kg)	constant	-	Edit		
	100000				
Solidus Temperature (k)	constant	•	Edit		
	1150			E	
Liquidus Temperature (k)	constant	•	Edit		
	1150				
1				•	
O	hange/Create	Delete Close		Help	

- d. Enter 680 j/kg-k for **Cp (Specific Heat)**.
- e. Enter 30 w/m-k for **Thermal Conductivity**.
- f. Enter 0.00553 kg/m-s for **Viscosity**.
- g. 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.

- h. Enter 1150 *K* for **Solidus Temperature**.
- i. Enter 1150 *K* for **Liquidus Temperature**.
- j. Click Change/Create and close the Create/Edit Materials dialog box.

24.4.6. Setting the Cell Zone Conditions

1. Set the cell zone conditions for the fluid (fluid).

\bigcirc Cell Zone Conditions → **\sqsubseteq** fluid → Edit...

💶 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 Embedded LES Reaction Source Terms Fixed Values Multiphase
This page is not applicable under current settings.
OK Cancel Help

- a. Select liquid-metal from the Material Name drop-down list.
- b. Click **OK** to close the **Fluid** dialog box.

24.4.7. Setting the Boundary Conditions

1. Set the boundary conditions for the inlet (inlet).

Boundary Cor	nditions	
Zone		
axis bottom-wall default-interior free-surface		
inlet		
outlet		
side-wall solid-wall		
Phase	Туре	ID
mixture	velocity-inlet	2
Inixtore	• Velocity Hillet •	2
Edit	Copy Profiles	
Parameters	Operating Conditions	
	Operating Conditions	
Display Mesh	Periodic Conditions	
Help		

$\diamondsuit Boundary \ Conditions \rightarrow \boxed{=} inlet \rightarrow Edit...$

💶 Velocity Inlet		X	
Zone Name			
inlet			
Momentum Thermal Radiation Species	s DPM Multiphase U	DS	
Velocity Specification Method	Magnitude, Normal to Boun	ndary 👻	
Reference Frame	Absolute	•	
Velocity Magnitude (m/s)	0.00101	constant 👻	
Supersonic/Initial Gauge Pressure (pascal)	0	constant 👻	
OK 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 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**).

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{=}$ outlet \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		—			
Zone Name					
outlet					
Momentum Thermal Radiation Species	Momentum Thermal Radiation Species DPM Multiphase UDS				
Velocity Specification Method	Components	▼			
Reference Frame	Absolute	•			
Supersonic/Initial Gauge Pressure (pascal)	0	constant 💌			
Axial-Velocity (m/s)	0.001	constant 🔹			
Radial-Velocity (m/s)	0	constant 💌			
Swirl-Velocity (m/s)	0	constant 🗸			
Swirl Angular Velocity (rad/s) 1					
OK Cancel Help					

a. Select **Components** from the **Velocity Specification Method** drop-down list.

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 🥃	3
Zone Name	
outlet	
Momentum Thermal Radiation Species DPM Multiphase UDS	_
Temperature (k) 500 constant	
OK 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			X
Zone Name bottom-wall			
Adjacent Cell Zone			
fluid	liation Species DPM Multiphase U	IDS Wall Film	
Thermal Conditions			
 Heat Flux Temperature 	Temperature (k)		constant 💌
Convection Radiation		Wall Thickness	(m) 0
 Mixed via System Coupling 	Heat Generation Rate (w/m3)	[-	constant •
Material Name aluminum	Contact Resistance (m2-k/w)	0	constant 🔹
	OK	Help	

- i. Select Temperature from 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).

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{\equiv}$ free-surface \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 Radiat	ion Species DPM Multiphase UDS Wall Film
Wall Motion Motion	
Stationary Wall Moving Wall	elative to Adjacent Cell Zone
Shear Condition	Marangoni Stress
 No Sip Specified Shear Specularity Coefficient Marangoni Stress 	Surface Tension Gradient (n/m-k) -0.00036
Wall Roughness	
Roughness Height (m)	constant 👻
Roughness Constant 0.5	constant v
	OK Cancel Help

a. Select Marangoni Stress from 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		x
Zone Name free-surface		
Adjacent Cell Zone		
fluid		
Momentum Thermal Rad	diation Species DPM Multiphase UDS Wall Film	
Thermal Conditions Heat Flux Temperature Convection Radiation Mixed via System Coupling Material Name aluminum	Heat Transfer Coefficient (w/m2-k) 100 constant Free Stream Temperature (k) 1500 constant Wall Thickness (m) 0 Heat Generation Rate (w/m3) 0 constant • Edit Contact Resistance (m2-k/w) 0 constant	• • •
	Cancel Help	

- 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).

$\clubsuit Boundary Conditions \rightarrow \blacksquare side-wall \rightarrow Edit...$

a. Click the **Thermal** tab.

💶 Wall			
vvali			×
Zone Name			
side-wall			
Adjacent Cell Zone			
fluid			
Momentum Thermal Rad	iation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
Heat Flux	Temperature (k)	1400	constant 👻
 Temperature 		Wall Thickness (
 Convection Radiation 		wai michless (end 6
Mixed	Heat Generation Rate (w/m3)	0	constant 👻
🔘 via System Coupling		-	constant
Material Name	Contact Resistance (m2-k/w)	0	constant 💌
aluminum	▼ Edit		
	OK	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**).

 $\clubsuit Boundary Conditions \rightarrow \blacksquare solid-wall \rightarrow Edit...$

💶 Wall	
💶 Wali	×
Zone Name	
solid-wall	
Adjacent Cell Zone	
fluid	
Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film	
Wall Motion Motion	
Stationary Wall Absolute Speed (rad/s) 1.0	
 Translational Rotational Components 	
Shear Condition	
 No Slip Specified Shear Specularity Coefficient Marangoni Stress 	
Wall Roughness	
Roughness Height (m) 0 constant v	
Roughness Constant 0.5 constant v	
OK Cancel Help	

a. Select **Moving Wall** from the **Wall Motion** group box.

The **Wall** dialog box will expand to show additional parameters.

b. Select **Rotational** in the lower box of the **Motion** group box.

The **Wall** dialog box will change 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 Zone			
fluid			
Momentum Thermal Rad	liation Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
Heat Flux	Temperature (k)	500	constant 👻
Temperature Convection Radiation		Wall Thickness (i	m) 0 (0
Mixed via System Coupling	Heat Generation Rate (w/m3)	P	constant 💌
Material Name	Contact Resistance (m2-k/w)	0	constant 💌
aluminum	▼ Edit		
	OK	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.

24.4.8. 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

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	Ŧ
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Options	
Default	
Hab	
Help	

- 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.

\bigcirc Solution Controls \rightarrow Equations...

Equations	
Equations	
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.

♀Solution Controls

Solution Controls	
Pseudo Transient Explicit Relaxation Factors	
Density	*
1	
Body Forces	
1	m.
Swirl Velocity	
0.75	
Energy	Ξ
0.75	
Liquid Fraction Update	
0.9	
	Ŧ
Default	
Equations Limits Advanced	
Help	

Retain the default values.

4. Enable the plotting of residuals during the calculation.

$\textcircled{} Monitors \rightarrow \overleftarrow{\sqsubseteq} Residuals \rightarrow Edit...$

Residual Monitors					×
Options	Equations				
✓ Print to Console	Residual	Monitor Ch	eck Convergence	Absolute Criteria	*
V Plot	energy	V	\checkmark	1e-06	
Window 1 Curves Axes					Ŧ
Iterations to Plot	, Residual Values			Convergence Crit	terion
1000	Normalize		5	absolute	•
Iterations to Store	Scale	l Scolo			
1000 A	Compute Loca	II SCAIE			
OK Plot	Renormalize	e Can	icel Hel	p	

- a. Ensure **Plot** is enabled in the **Options** group box.
- b. Click **OK** to accept the remaining default settings and close the **Residual Monitors** dialog box.
- 5. Initialize the solution.

Over Solution Initialization

Solution Initialization
Initialization Methods Hybrid Initialization Standard Initialization
More Settings Initialize Patch Reset DPM Sources Reset Statistics
Help

a. Retain the default of Hybrid Initialization from the Initialization Methods group box.

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.

Define \rightarrow **Custom Field Functions...**

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 Ω 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 Ω is included for demonstration purposes.

💶 Custom Field Function Calculator		
Definition radial-coordinate * 1 + - X / y^x ABS Select Operand Field Functions from INV sin cos tan In log 10 Field Functions 0 1 2 3 4 SQRT Mesh ▼ 5 6 7 8 9 CE/C Radial Coordinate ▼ () PI e DEL Select Select New Function Name omegar omegar Omegar Omegar Omegar		
Define Manage Close Help		

- a. Select Mesh... and Radial Coordinate from the Field Functions drop-down lists.
- 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 \times button on the calculator pad.
- d. Click the **1** button.
- e. Enter omegar for New Function Name.
- f. Click Define.

Note

To check the function definition or delete the custom field function, click **Manage...** to open the **Field Function Definitions** dialog box. Then select **omegar** from the **Field Functions** selection list to view the function definition.

- g. Close the Custom Field Function Calculator dialog box.
- 7. Patch the pull velocities.

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.

Patch		X
Reference Frame Relative to Cell Zone Absolute Variable Axial Velocity Radial Velocity Swirl Velocity Temperature Axial Pull Velocity Radial Pull Velocity Swirl Pull Velocity Swirl Pull Velocity	Value (m/s) 0.001 Use Field Function Field Function Omegar	Zones to Patch
	Patch Close Help	

- a. Select Axial Pull Velocity from the Variable selection list.
- b. Enter 0.001 m/s for Value.
- c. Select **fluid** from the **Zones to Patch** selection list.
- d. Click Patch.

You have just patched the axial pull velocity. Next you will patch the swirl pull velocity.

Patch			×
Reference Frame Relative to Cell Zone Absolute Variable Axial Velocity Radial Velocity Swirl Velocity Temperature Axial Pull Velocity Radial Pull Velocity Swirl Pull Velocity	Value (m/s) 0 Value Tield Function Field Function Omegar	Zones to Patch) = h) =
	Patch Close Help		

e. Select Swirl Pull Velocity from the Variable selection list.

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.

Run Calculation

Run Calculation
Check Case Preview Mesh Motion
Pseudo Transient Options
Fluid Time Scale
Time Step Method Pseudo Time Step (s)
User Specified Automatic
Solid Time Scale
Time Step Method Pseudo Time Step (s)
User Specified Automatic
Number of Iterations Reporting Interval 20 1
Profile Update Interval
Data File Quantities Acoustic Signals
Calculate
Help

- a. 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. Display filled contours of temperature (Figure 24.3: Contours of Temperature for the Steady Conduction Solution (p. 1022)).

Contours		×
Options	Contours of	
✓ Filled	Temperature	•
Vode Values Global Range	Static Temperature	•
Auto Range	Min Max	
Clip to Range	0 0	
Draw Mesh	Surfaces	
	axis	•
Levels Setup	bottom-wall default-interior	=
	free-surface	
	inlet	-
Surface Name Pattern	New Surface 🔻	
Match	Surface Types	
	axis	*
	clip-surf	
	exhaust-fan fan	
	-	*

Graphics and Animations → $\overline{\equiv}$ Contours → Set Up...

- a. Enable the **Filled** option.
- b. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.
- c. Click **Display** (Figure 24.3: Contours of Temperature for the Steady Conduction Solution (p. 1022)).

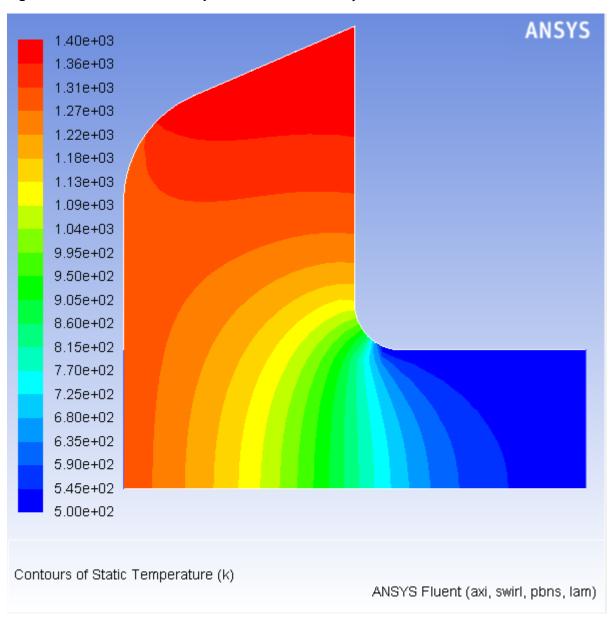


Figure 24.3: Contours of Temperature for the Steady Conduction Solution

11. Display filled contours of temperature to determine the thickness of mushy zone.

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

Contours	×	
Options	Contours of	
✓ Filled	Temperature 👻	
 Node Values Global Range 	Static Temperature 🗸	
Auto Range	Min (k) Max (k)	
Clip to Range	1100 1200	
Draw Mesh	Surfaces	
	axis	
Levels Setup	bottom-wall	
	idetault-interior	
20 🔺 1		
	inlet 👻	
Surface Name Pattern	New Surface	
Match	Surface Types	
	axis	
	clip-surf	
	exhaust-fan	
	fan 👻	
Display Compute Close Help		

a. Disable Auto Range in the Options group box.

The *Clip to Range* option will automatically be enabled.

- b. Enter 1100 for Min and 1200 for Max.
- c. Click **Display** (See Figure 24.4: Contours of Temperature (Mushy Zone) for the Steady Conduction Solution (p. 1024)) and close the **Contours** dialog box.

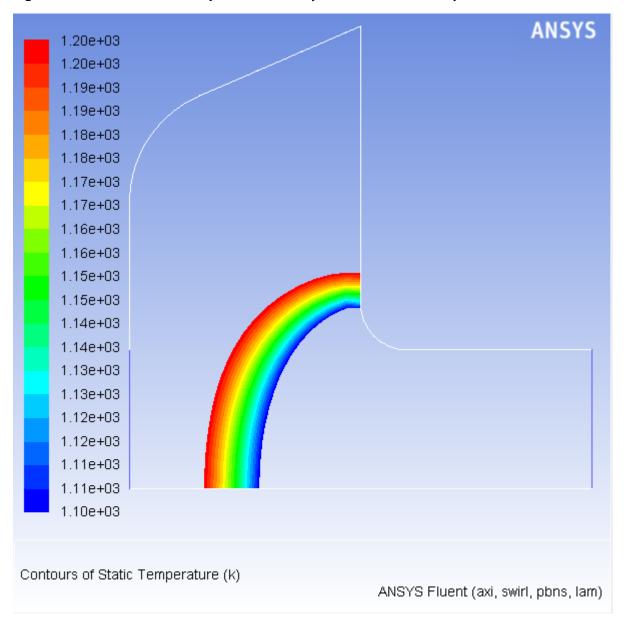


Figure 24.4: Contours of Temperature (Mushy Zone) for the Steady Conduction Solution

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...

24.4.9. 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.

∲General

General	
Mesh Scale Display	Check Report Quality
Solver	
Type ● Pressure-Based ○ Density-Based	Velocity Formulation Absolute Relative
Time Steady Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Gravitational Acceleration	n
X (m/s2) -9.81	P
Y (m/s2)	e
Z (m/s2)	[²
Help	

- a. Select **Transient** from the **Time** list.
- 2. Set the solution parameters.

Solution Methods

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	
First Order Implicit 👻	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
High Order Term Relaxation Options	
Default	
Help	

- 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.

\clubsuit Solution Controls \rightarrow Equations...

Equations		×
Equations		
Flow		
Swirl Velocity Energy		
OK Default	Cancel	Help
		<u> </u>

- 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.

Over Solution Controls

Solution Controls				
Flo	ow Courant N	umber		
2	200			
	Explicit Relaxa	ation Factors		
	Momentum			
	Pressure	0.75		
Under-Rel	axation Facto	rs		
Density			*	
1				
Body Fo	rces			
1	Body Forces			
Swirl Vel	ocity			
0.9	Swirl Velocity 0.9			
Liquid Fr	action Update	e		
0.1	action optice	<u> </u>		
Energy				
1				
			Ŧ	
Default				
Equation	s Limits.	Advanced		
Help				

- 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.

CRun Calculation

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 Time Sampling Interval	Sampling Options
Max Iterations/Time Step	Reporting Interval
20	1
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
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.

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

- a. Ensure that **Temperature...** and **Static Temperature** are selected from the **Contours of** drop-down lists.
- b. Click **Display** (See Figure 24.5: Contours of Temperature at t=0.2 s (p. 1029)).

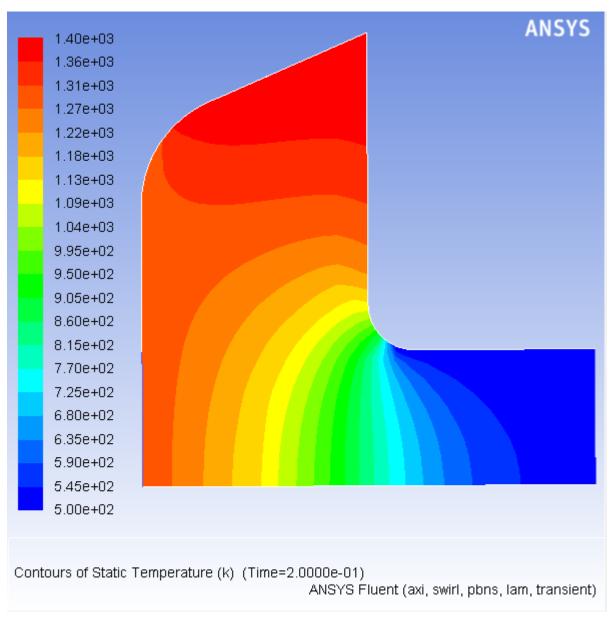


Figure 24.5: Contours of Temperature at t=0.2 s

8. Display contours of stream function (Figure 24.6: Contours of Stream Function at t=0.2 s (p. 1030)).



- a. Disable Filled in the Options group box.
- b. Select Velocity... and Stream Function from the Contours of drop-down lists.
- c. Click **Display**.

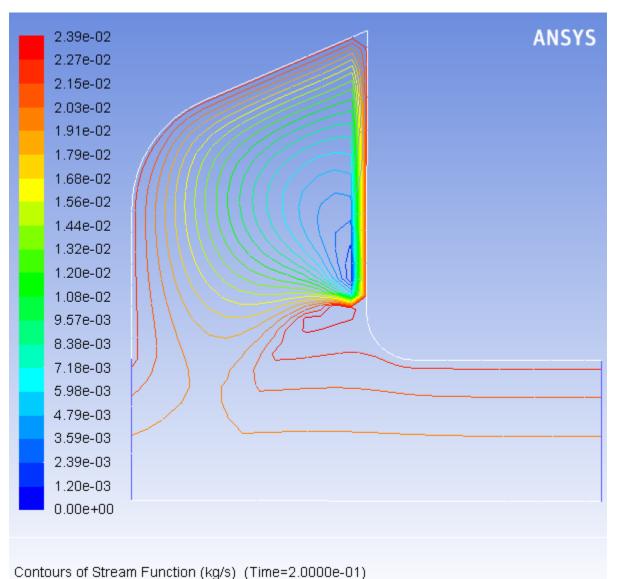


Figure 24.6: Contours of Stream Function at t=0.2 s

ANSYS Fluent (axi, swirl, pbns, lam, transient)

As shown in Figure 24.6: Contours of Stream Function at t=0.2 s (p. 1030), the liquid is beginning to circulate in a large eddy, driven by natural convection and Marangoni convection on the free surface.

9. Display contours of liquid fraction (Figure 24.7: Contours of Liquid Fraction at t=0.2 s (p. 1031)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

- a. Enable **Filled** in the **Options** group box.
- b. Select Solidification/Melting... and Liquid Fraction from the Contours of drop-down lists.
- c. Click **Display** and close the **Contours** dialog box.

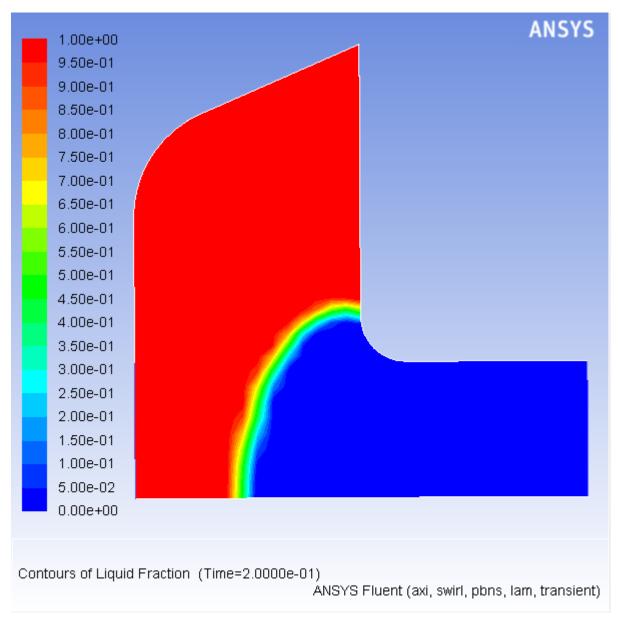


Figure 24.7: Contours of Liquid Fraction at t=0.2 s

The liquid fraction contours show the current position of the melt front. Note that in Figure 24.7: Contours of Liquid Fraction at t=0.2 s (p. 1031), the mushy zone divides the liquid and solid regions roughly in half.

10. Continue the calculation for 48 additional time steps.

CRun Calculation

- a. Enter 48 for Number of Time Steps.
- b. Click Calculate.

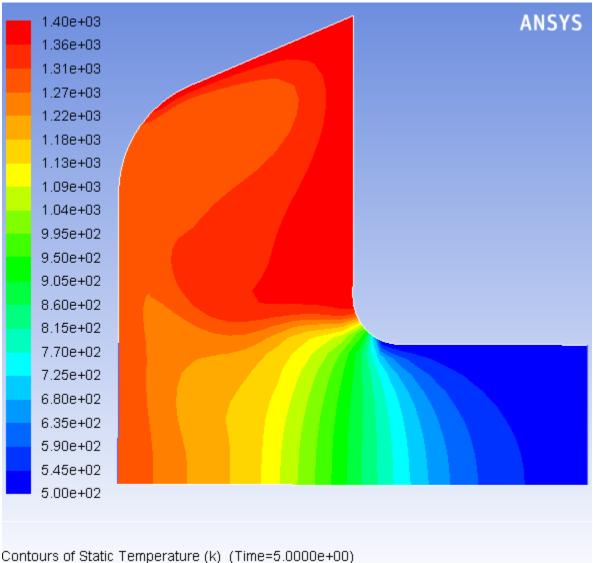
After a total of 50 time steps have been completed, the elapsed time will be 5 seconds.

11. Display filled contours of the temperature after 5 seconds (Figure 24.8: Contours of Temperature at t=5 s (p. 1032)).

Graphics and Animations → $\overline{\Xi}$ Contours → Set Up...

- 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. Click **Display**.





ANSYS Fluent (axi, swirl, pbns, lam, transient)

As shown in Figure 24.8: Contours of Temperature at t=5 s(p. 1032), 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 (i.e., 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 24.9: Contours of Stream Function at t=5 s (p. 1033)).

Graphics and Animations → \equiv Contours → Set Up...

- a. Disable **Filled** in the **Options** group box.
- b. Select Velocity... and Stream Function from the Contours of drop-down lists.
- c. Click **Display**.

As shown in Figure 24.9: Contours of Stream Function at t=5 s (p. 1033), the flow has developed more fully by 5 seconds, as compared with Figure 24.6: Contours of Stream Function at t=0.2 s (p. 1030) after 0.2 seconds. The main eddy, driven by natural convection and Marangoni stress, dominates the flow.

To examine the position of the melt front and the extent of the mushy zone, you will plot the contours of liquid fraction.

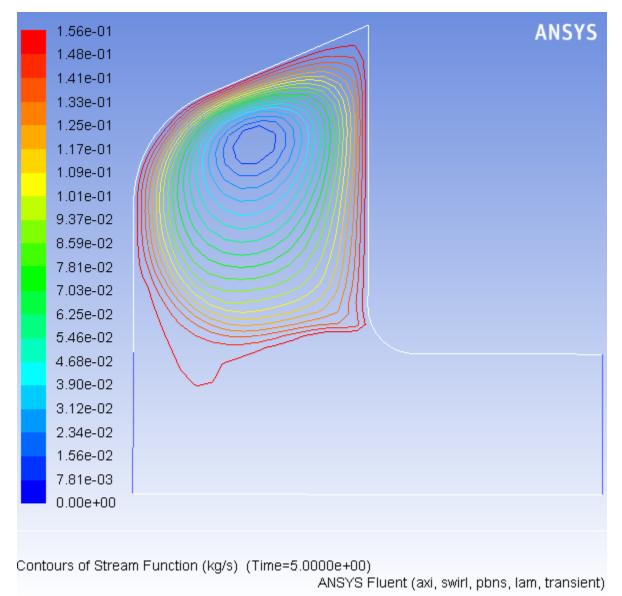


Figure 24.9: Contours of Stream Function at t=5 s

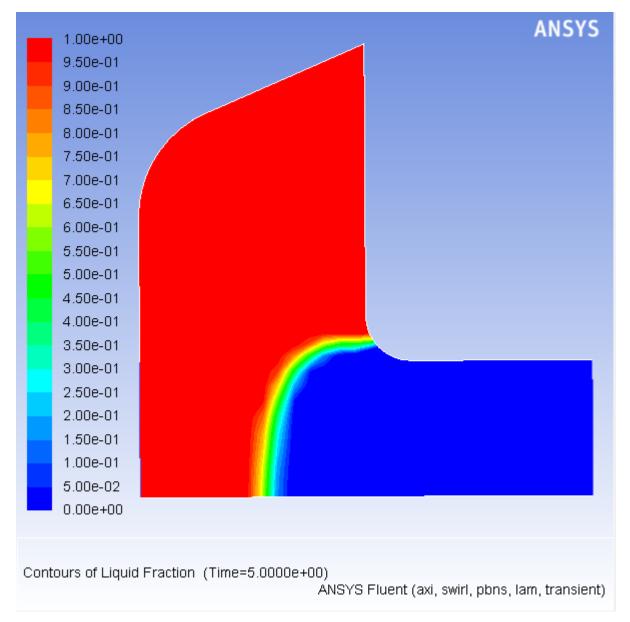
13. Display filled contours of liquid fraction (Figure 24.10: Contours of Liquid Fraction at t=5 s (p. 1034)).



- a. Enable **Filled** in the **Options** group box.
- b. Select Solidification/Melting... and Liquid Fraction from the Contours of drop-down lists.
- c. Click **Display** and close the **Contours** dialog box.

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 24.10: Contours of Liquid Fraction at t=5 s(p. 1034)).

Figure 24.10: 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...

24.5. 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 Modeling Solidification and Melting in the User's Guide.

24.6. 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. 123).

Chapter 25: Using the Eulerian Granular Multiphase Model with Heat Transfer

This tutorial is divided into the following sections:

25.1. Introduction25.2. Prerequisites25.3. Problem Description25.4. Setup and Solution25.5. Summary25.6. Further Improvements25.7. References

25.1. 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.

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. 123)

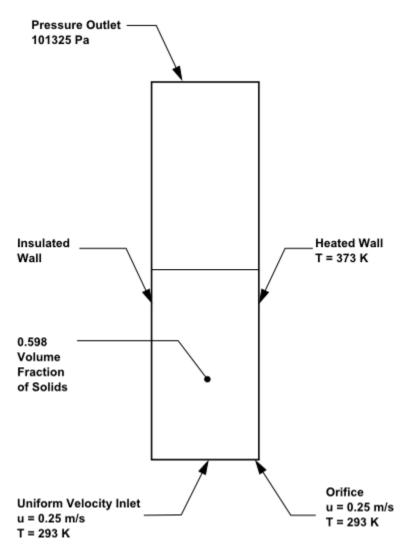
and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

In order to complete the steps to compile the UDF, you will need to have a working C compiler installed on your machine.

25.3. 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 25.1: Problem Schematic (p. 1038).

Figure 25.1: Problem Schematic



25.4. Setup and Solution

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

25.4.1. Preparation 25.4.2. Mesh 25.4.3. General Settings 25.4.4. Models 25.4.5. UDF 25.4.6. Materials 25.4.7. Phases 25.4.8. Boundary Conditions 25.4.9. Solution

25.4.10. Postprocessing

25.4.1. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip eulerian_granular_heat_R150.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 Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Run in Serial by selecting Serial under Processing Options.
- 11. Ensure that **Setup 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.

25.4.2. Mesh

1. Read the mesh file fluid-bed.msh.

```
\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}
```

As ANSYS Fluent reads the mesh file, it will report the progress in the console.

25.4.3. General Settings

General

General	
Mesh Scale Display	Check Report Quality
Solver	
Type ● Pressure-Based ○ Density-Based	Velocity Formulation O Absolute O Relative
Time ◉ Steady ◯ Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

1. Check the mesh.

$\mathbf{O}_{General} \rightarrow \mathbf{Check}$

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.

2. Examine the mesh (Figure 25.2: Mesh Display of the Fluidized Bed (p. 1041)).

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.

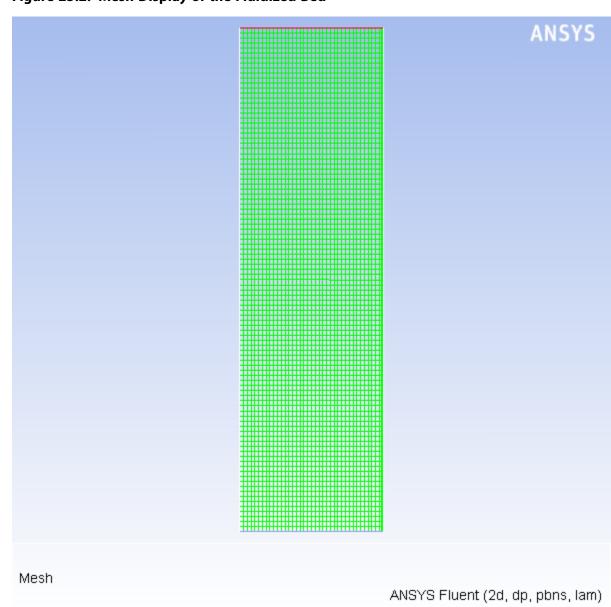


Figure 25.2: Mesh Display of the Fluidized Bed

3. Enable the pressure-based transient solver.

General

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.
- 4. Set the gravitational acceleration.

```
\mathbf{\mathbf{\dot{\mathbf{\nabla}}}} \mathbf{General} \rightarrow \mathbf{\mathbf{\overline{M}}} \mathbf{\mathbf{Gravity}}
```

General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Type Pressure-Based	Velocity Formulation Absolute
 Density-Based 	© Relative
Time	2D Space
 Steady Transient 	 Planar Axisymmetric
	Axisymmetric Swirl
Gravity	Units
Gravitational Acceleratio	n
X (m/s2)	e
Y (m/s2) -9.81	P
Z (m/s2)	9
<u>e</u>	
Help	

a. Enter -9.81 m/s² for the **Gravitational Acceleration** in the **Y** direction.

25.4.4. Models

1. Enable the Eulerian multiphase model for two phases.



Multiphase Model	—
Model Off Volume of Fluid Mixture Eulerian Wet Steam Eulerian Parameters	Number of Eulerian Phases
Dense Discrete Phase Model Boiling Model Multi-Fluid VOF Model	
Volume Fraction Parameters Scheme © Explicit @ Implicit	
OK Cance	l Help

- a. Select Eulerian from the Model list.
- b. Click **OK** to close the **Multiphase Model** dialog box.
- 2. Enable heat transfer by enabling the energy equation.

 $\mathbf{O} \mathsf{Models} \rightarrow \mathbf{E} \mathsf{Energy} \rightarrow \mathsf{Edit...}$

Energy	-X
Energy	
Energy Equation	
OK Cancel	Help

- a. Enable **Energy Equation**.
- b. Click **OK** to close the **Energy** dialog box.

An Information dialog box will open. Click OK to close the Information dialog box.

3. Retain the default laminar viscous model.

 $\mathbf{O} \mathsf{Models} \to \mathbf{E} \mathsf{Viscous} \to \mathsf{Edit...}$

💶 Viscous Model 🛛 🗾
Model
 ② Laminar ○ k-epsilon (2 eqn) ○ k-omega (2 eqn) ○ Reynolds Stress (5 eqn)
Options
Viscous Heating
OK Cancel Help

Experiments have shown negligible three-dimensional effects in the flow field for the case modeled, suggesting very weak turbulent behavior.

25.4.5. UDF

1. Compile the user-defined function, conduct.c, that will be used to define the thermal conductivity for the gas and solid phase.

Compiled UDFs			x
Source Files conduct.c		Header Files	
Add Delete		Add Delete	
Library Name libudf		Build	
	Load Car	ncel Help	

Define \rightarrow **User-Defined** \rightarrow **Functions** \rightarrow **Compiled...**

- 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.

25.4.6. Materials

1. Modify the properties for air, which will be used for the primary phase.

\mathbf{O} Materials \rightarrow	air →	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
Chemical Formula	fluid FLUENT Fluid Materials	Name Chemical Formula
	air	 FLUENT Database
	Mixture	User-Defined Database
	none	v
Properties		
Density (kg/m3)	constant 👻 Edit	
	1.2	
Cp (Specific Heat) (j/kg-k)	constant	
	994	
Thermal Conductivity (w/m-k)	user-defined	
	conduct_gas::libudf	
Viscosity (kg/m-s)	constant	
	1.7894e-05	
Cha	ige/Create Delete Close Help	

- a. Enter 1.2 kg/m^3 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).

$\textcircled{} Materials \rightarrow \fbox{} air \rightarrow Create/Edit...$

Create/Edit Materials		
Name	Material Type	Order Materials by
Chemical Formula	fluid FLUENT Fluid Materials	Chemical Formula
	solids	FLUENT Database
	Mixture	User-Defined Database
	none	*
Properties		
Density (kg/m3)	constant Edit	
	2660	
Cp (Specific Heat) (j/kg-k)	constant Edit E	
	737	
Thermal Conductivity (w/m-k)	user-defined Edit	
	conduct_solid::libudf	
Viscosity (kg/m-s)	constant 💌 Edit	
	1.7894e-05	
,		
Cha	nge/Create Delete Close Help	

- a. Enter solids for Name.
- b. Enter 2660 kg/m³ 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.

25.4.7. Phases

⇔Phases

Phases	
Phases	
phase-1 - Primary Phase	
phase-2 - Secondary Phase	
Edit Interaction	ID 2
\square	
Help	

1. Define air as the primary phase.

♦ Phases $\rightarrow \stackrel{\frown}{=}$ phase-1 - Primary Phase \rightarrow Edit...

Primary Phase	×
Name	
air	
Phase Material air	
OK 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.

Phases → $\overline{\equiv}$ phase-2 - Secondary Phase → Edit...

Secondary Phase		×
Name		
solids		
Phase Material solids	▼ Edit	
🗸 Granular		
Packed Bed		
Granular Temperature Model	1	
 Phase Property Partial Differential Equation 		
Properties		
Solids Pressure (pascal)	lun-et-al 👻	Edit
Radial Distribution	lun-et-al 👻	Edit
Elasticity Modulus (pascal)	derived 👻	Edit
		=
Packing Limit	constant 👻	Edit
	0.6	
1		Ψ.
OK	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.

$\mathbf{\hat{\mathbf{v}}}$ Phases \rightarrow Interaction...

Phase Interaction	1				
Virtual Mass					
Drag Lift Wa	all Lubrication Turbulent	Dispersion Turbulence Interaction Collision	s Slip Heat Mas	s Reactions Surface Tension	Discretization Interfacial Area
Drag Coefficient					
Drag Modification	on		-	L	
solids	air	syamlal-obrien	▼ Edt		
1		1		7	
		ОК Са	ncel Help		
		OK Ca	ncel Heb		

- a. Select syamlal-obrien from the Drag Coefficient drop-down list.
- b. Click the Heat tab, and select gunn from the Heat Transfer Coefficient drop-down list.

	Phase Interaction						
	Virtual Mass						
0	yrag Lift Wall	Lubrication Turbuler	t Dispersion Turbulence Intera	ction Collisions Slip	Heat Ma	ass Reactions Surface Tensio	n Discretization Interfacial Area
H	eat Transfer Coefficie	ent					
	solids	air	gunn	•	Edit	^	
						-	
				OK Cancel I	telp		

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.

25.4.8. Boundary Conditions

For this problem, you need to set the boundary conditions for all boundaries.

Boundary Conditions

Boundary Conditions

Zone		
default-interior		
poutlet		
v_jet		
v_uniform		
wall_hot wall_ins		
wai_ins		
Phase	Туре	ID
air	✓ velocity-inlet ✓	5
Edit	Copy Profiles	
Deservations		
Parameters	Operating Conditions	
Display Mesh	Periodic Conditions	
Help		

1. Set the boundary conditions for the lower velocity inlet (**v_uniform**) for the primary phase.

$\textcircled{P} Boundary \ Conditions \rightarrow \fbox{v_uniform}$

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		Phase
v_uniform		air
Momentum Thermal Radia		
Reference Frame	L	
		
Velocity Magnitude (m/s)	0.25	constant 👻
	OK Cancel He	þ

- 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.

\bigcirc Boundary Conditions $\rightarrow \equiv v_{uniform}$

- a. Select **solids** from the **Phase** drop-down list.
- b. Click the Edit... button to open the Velocity Inlet dialog box.

Velocity Inlet	×
Zone Name v_uniform	Phase solids
Momentum Thermal Radiation Species DPM Multiphase Temperature (k) 293 constant	UDS
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.

• Boundary Conditions $\rightarrow \equiv v_{jet}$

- 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 v_jet		Phase
Momentum Thermal Radia	tion Species DPM Multipha	
Reference Frame		
Velocity Magnitude (m/s)	0.25 con	stant 🔹
	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.

• Boundary Conditions $\rightarrow \equiv v_{jet}$

- a. Select solids from the Phase drop-down list.
- b. Click the Edit... button to open the Velocity Inlet dialog box.

Velocity Inlet	x					
Zone Name	Phase					
v_jet	solids					
	Momentum Thermal Radiation Species DPM Multiphase 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.

◆Boundary Conditions → Èpoutlet

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.

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{=}$ poutlet

- a. Select air from the Phase drop-down list.
- b. Click the Edit... button to open the Pressure Outlet dialog box.

Pressure Outlet	×
Zone Name poutlet	Phase air
Momentum Thermal Radiation Species DPM Multiphase Backflow Total Temperature (k) 293 constant	
OK Cancel Help	

- i. Click the Thermal tab and 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.



- a. Select **solids** from the **Phase** drop-down list.
- b. Click the **Edit...** button to open the **Pressure Outlet** dialog box.

Pressure Outlet	×			
Zone Name poutlet	Phase solids			
Backflow Total Temperature (k) 293 constant				
OK Cancel Help				

- i. Click the Thermal tab and enter 293 K for the Backflow Total Temperature.
- ii. Click the Multiphase tab and 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.

\bigcirc Boundary Conditions $\rightarrow \stackrel{\frown}{=} wall_hot$

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		Phase	
wall_hot		mixture	
Adjacent Cell Zone		all self self parts in the	
fluid			
Momentum Thermal Radiat	tion Species DPM Multiphase	UDS Wall Film	
Thermal Conditions			
C Heat Flux	Temperature (k	373	constant 👻
 Temperature Convection 		Wall Thickness	(m) 0
© Radiation			P
C Mixed	Heat Generation Rate (w/m3	0	constant 💌
via System Coupling			
Material Name			
aluminum	▼ Edit		
	OK	el Help	

- i. Click the **Thermal** tab.
 - A. Select Temperature from the Thermal Conditions list.
 - B. Enter 373 K for **Temperature**.
- ii. Click **OK** to close the **Wall** dialog box.
- 9. Set the boundary conditions for the heated wall (wall_hot) for the primary phase.

\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	Phase
wall_hot	air
Adjacent Cell Zone	
fluid	
Momentum Thermal Radiation Species DPM Multiphase	UDS Wall Film
 No Slip Specified Shear Specularity Coefficient Marangoni Stress 	
OK Cancel 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 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) for the primary phase.



For the adiabatic wall, you will retain the default thermal conditions for the mixture (zero heat flux), and the default momentum conditions (no slip) for both phases.

- a. Select air from the Phase drop-down list.
- b. Click the **Edit...** button to open the **Wall** dialog box.

💶 Wall	— ×
Zone Name	Phase
wall_ins	air
Adjacent Cell Zone	
fluid	
Momentum Thermal Radiation Species DPM Multiphase	UDS Wall Film
Shear Condition	
No Slip	
 Specified Shear Specularity Coefficient 	
Marangoni Stress	
OK Cancel Help	

- c. Retain the default **No Slip** condition and click **OK** to close the **Wall** dialog box.
- 12. Set the boundary conditions for the adiabatic wall (**wall_ins**) for the secondary phase same as that of the primary phase.

• Boundary Conditions $\rightarrow \stackrel{\frown}{=} wall_{ins}$

For the secondary phase, you will retain the default no slip condition as for the primary phase.

25.4.9. Solution

1. Select the second order implicit transient formulation.

Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Phase Coupled SIMPLE 🔹	
Spatial Discretization	
Gradient	A
Least Squares Cell Based 🔹	
Momentum	
Second Order Upwind	
Volume Fraction	=
QUICK 🗸	
Energy	
QUICK 🔹	
	Ŧ
Transient Formulation	
Second Order Implicit 🔹	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
High Order Term Relaxation Options	
Default	
Help	

- 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

Solution Controls	
Under-Relaxation Factors	
Pressure	<u>.</u>
0.5	
Density	
1	
Body Forces	Ξ
1	
Momentum	
0.2	
Volume Fraction	
0.5	÷
Default	
Equations Limits Advanced	
Help	

- a. Enter 0.5 for **Pressure**.
- b. Enter 0.2 for Momentum.
- 3. Ensure that the plotting of residuals is enabled during the calculation.

♦ Monitors → 🔄 Residuals → Edit...

4. Define a custom field function for the heat transfer coefficient.

Define → **Custom Field Functions...**

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.

- 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} .

Definition air-thermal-conductivity-lam * air-vof + solids-thermal-conductivity-lam * solids-vof + - X / Y^X ABS Select Operand Field Functions from INV sin cos tan ln log10 0 1 2 3 4 SQRT Field Functions Phases Volume fraction Phase solids solids New Function Name k_mix

- 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 ave_htc.

Custom Field Function Calculator Definition - k_mix * (t_mix - 373) / (58.5 * 10 ^ (- 6)) / 80	×	
+ - X / y^x ABS Select Operand Field Functions from INV sin cos tan In log 10 0 1 2 3 4 SQRT 5 6 7 8 9 CE/C () PI e DEL Phase mixture Select New Function Name ave_htc		
Define Manage Close Help		

- i. Click the subtraction symbol in the calculator pad.
- ii. Select **Custom Field Functions...** and **k_mix** from the **Field Functions** drop-down lists.
- 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.

Surface → Point...

Point Surface	—		
Options Point Tool Reset	Coordinates x0 (m) 0.28494 y0 (m) 0.24 z0 (m) 0		
Select Point with Mouse			
New Surface Name			
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. Define a surface monitor for the heat transfer coefficient.

• Monitors (Surface Monitors) \rightarrow Create...

Surface Monitor	×	
Name	Report Type	
surf-mon-1	Facet Average 🔹	
Options	Field Variable	
Print to Console	Custom Field Functions 🔻	
V Plot	ave_htc	
Window	Phase	
2 Curves Axes	mixture 🔹	
Vite	Surfaces	
File Name	default-interior	
htc-024.out	poutlet	
nic-024.001	v_jet	
X Axis	v_uniform	
Flow Time	wall_hot	
	wall_ins	
Get Data Every	y=0.24	
1 Time Step		
Average Over(Time Steps)	New Surface 🔻	
1		
OK Cancel Help		

- a. Enable **Plot**, and **Write** for **surf-mon-1**.
- b. Enter htc-024.out for File Name.
- c. Select Flow Time from the X Axis drop-down list.
- d. Select Time Step from the Get Data Every drop-down list.
- e. Select Facet Average from the Report Type drop-down list.
- f. Select Custom Field Functions... and ave_htc from the Field Variable drop-down lists.
- g. Select y=0.24 from the Surfaces selection list.
- h. Click OK to close the Surface Monitor dialog box.
- 7. Initialize the solution.

CSolution Initialization

Solution Initialization	
Initialization Methods	
Hybrid InitializationStandard Initialization	
Compute from	_
all-zones	•
Reference Frame	
 Relative to Cell Zone Absolute 	
Initial Values	
Gauge Pressure (pascal)	1
0	
air X Velocity (m/s)	
0	
air Y Velocity (m/s)	=
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	
Help	

- 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.

$\textbf{Adapt} \rightarrow \textbf{Region...}$

This register is used to patch the initial volume fraction of solids in the next step.

Region Adaption		
Options	Input Coordinates	
Inside	X Min (m)	X Max (m)
Outside	0	0.3
Shapes	Y Min (m)	Y Max (m)
Quad	0	0.5
 Circle Cylinder 	Z Min (m)	Z Max (m)
	0	0
Manage		
Controls	0	
Select Points with Mouse		
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.

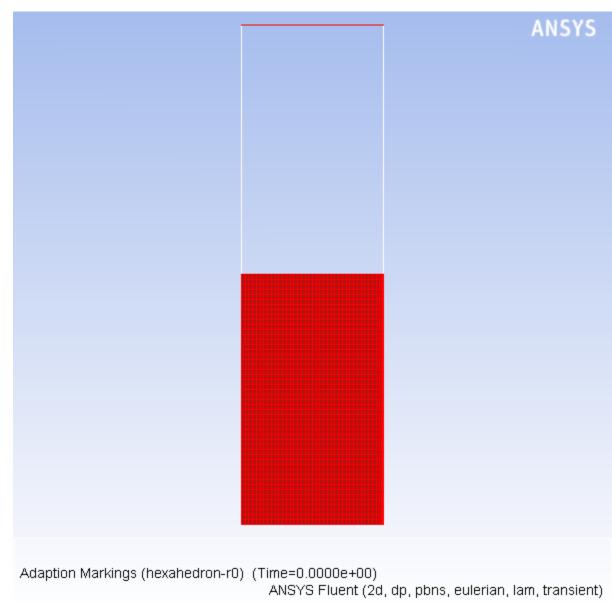


Figure 25.3: Region Marked for Patching

d. Close the **Region Adaption** dialog box.

9. Patch the initial volume fraction of solids in the lower half of the fluidized bed.

Solution Initialization → Patch...

Patch		—
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 It_mix k_mix ave_htc	Zones to Patch
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 25.4: Initial Volume Fraction of Granular Phase (solids). (p. 1067)).

• Graphics and Animations $\rightarrow \equiv$ Contours \rightarrow Set Up...

- 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.

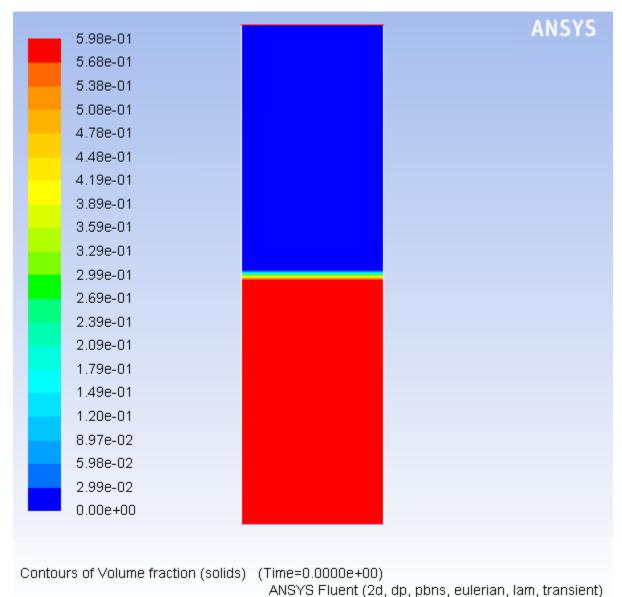


Figure 25.4: Initial Volume Fraction of Granular Phase (solids).

11. Save the case file (fluid-bed.cas.gz).

$\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}$

12. Start calculation.

CRun Calculation

Run Calculation		
Check Case	Preview Mesh Motion	
Time Stepping Method	Time Step Size (s)	
Fixed 🔹	0.00015	
Settings	Number of Time Steps	
	12000	
Options		
Extrapolate Variables Data Sampling for Time Statistics Sampling Interval Sampling Options Time Sampled (s) 0		
Max Iterations/Time Step	Reporting Interval	
Profile Update Interval		
Data File Quantities	Acoustic Signals	
Calculate		
Help		

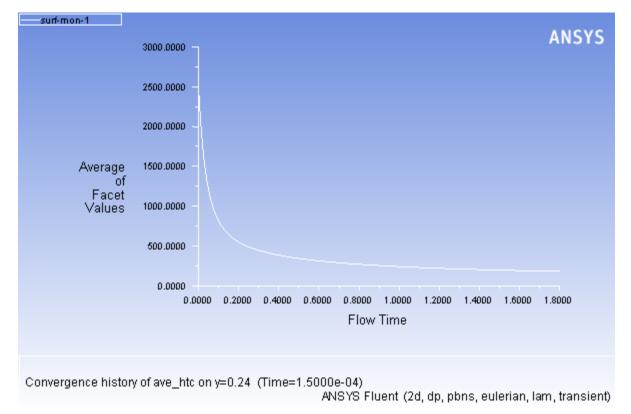
- a. Set 0.00015 for Time Step Size(s).
- b. Set 12000 for Number of Time Steps.
- c. Enter 50 for Max Iterations/Time Step.
- d. Click Calculate.

Note

If you do not want to wait for the solution to run the full 12000 time steps you can stop the calculation and read in the completed case/data files from the solution-files subdirectory in the eulerian_granular_heat.zip file.

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 25.5: 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...

25.4.10. Postprocessing

1. Display the pressure field in the fluidized bed (Figure 25.6: Contours of Static Pressure (p. 1071)).



Contours			
Options		Contours of	
Filled		Pressure	-
V Node Values Global Range		Static Pressure	•
Auto Range		Phase	
Clip to Range		mixture 🗸	
Draw Profiles		Min (pascal)	Max (pascal)
Draw Mesh		-0.002768115	7755.541
Levels Setup		Surfaces	
20 🛋 1		default-interior	*
		poutlet	=
		v_jet v_uniform	
Surface Name Pattern		wall_hot	
M	atch		*
		New Surface 💌	
		Surface Types	
		axis	A
		clip-surf	
		degassing	
		exhaust-fan	Ψ
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**.

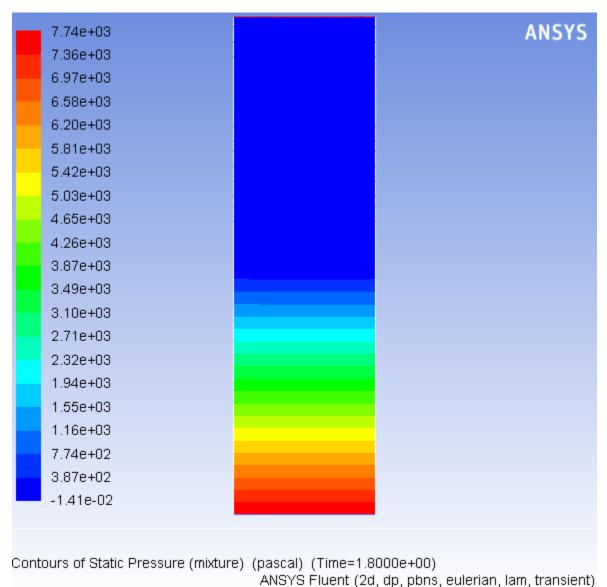


Figure 25.6: Contours of Static Pressure

Note the build-up of static pressure in the granular phase.

- 2. Display the volume fraction of solids (Figure 25.7: Contours of Volume Fraction of Solids (p. 1072)).
 - 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.

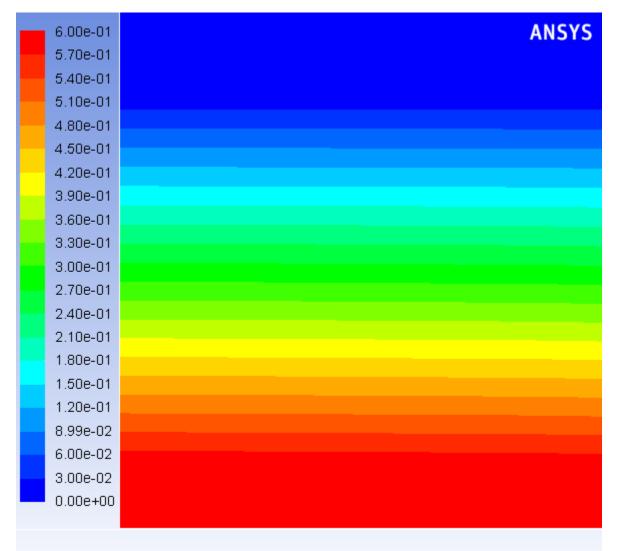


Figure 25.7: Contours of Volume Fraction of Solids

Contours of Volume fraction (solids) (Time=1.8000e+00) ANSYS Fluent (2d, dp, pbns, eulerian, lam, transient)

25.5. 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].

25.6. 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. 123).

25.7. 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 26: Postprocessing

This tutorial is divided into the following sections:

26.1. Introduction26.2. Prerequisites26.3. Problem Description26.4. Setup and Solution26.5. Summary

26.1. Introduction

This tutorial demonstrates the postprocessing capabilities of ANSYS 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, is 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.
- Display results on successive slices of the domain.
- Display pathlines.
- Plot quantitative results.
- Overlay and "explode" a display.
- Annotate the display.

26.2. 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. 123)

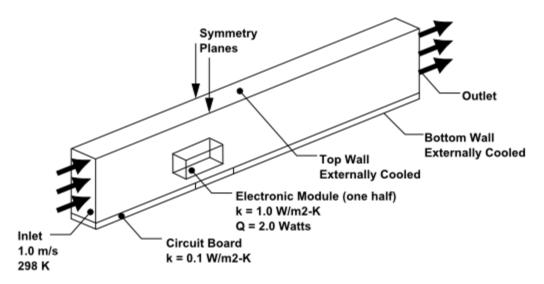
and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

26.3. Problem Description

The problem considered is shown schematically in Figure 26.1: Problem Specification (p. 1076). 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 / m²-K. The circuit board conductivity is assumed to be one order of magnitude lower: 0.1 W / m²-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 26.1: Problem Specification



26.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial: 26.4.1. Preparation 26.4.2. Reading the Mesh 26.4.3. Manipulating the Mesh in the Viewer 26.4.4. Adding Lights
26.4.5. Creating Isosurfaces
26.4.6. Generating Contours
26.4.7. Generating Velocity Vectors
26.4.8. Creating Animation
26.4.9. Displaying Pathlines
26.4.10. Overlaying Velocity Vectors on the Pathline Display
26.4.11. Creating Exploded Views
26.4.12. Animating the Display of Results in Successive Streamwise Planes
26.4.13. Generating XY Plots
26.4.14. Creating Annotation
26.4.15. Saving Hardcopy Files
26.4.16. Generating Volume Integral Reports

26.4.1. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip the postprocess_R150.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 ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the *Fluent Getting Started Guide*.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, 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.

26.4.2. Reading the Mesh

1. Read in the case and data files chip.cas.gz and chip.dat.gz.

```
File \rightarrow Read \rightarrow Case & Data...
```

When you select the case file, ANSYS Fluent will read the data file automatically.

26.4.3. Manipulating the Mesh in the Viewer

1. Display the mesh surfaces **board-top** and **chip**.

$\mathbf{\mathbf{\dot{\mathbf{\nabla}}}} \mathbf{General} \rightarrow \mathbf{Display...}$

💶 Mesh Displa	у		×
Options	Edge Type	Surfaces	
Nodes	● All	board-ends	
Edges Edges	 Feature Outline 	board-sym board-top	=
Partitions	Oddine	bottom-wall	
		chip	
	Feature Angle	chip-bottom	
0	20	chip-sym	-
Surface Name Pa	ttern Match	New Surface 💌	
		Surface Types	
Outline	ior	axis	
		clip-surf	
Adjacency		exhaust-fan	
fan v			
Display Colors Close Help			

- a. Retain the default selection of Edges in the Options group box.
- b. Deselect all surfaces and select **board-top** and **chip** from the **Surfaces** selection list.

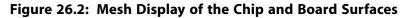
To deselect all surfaces click the far-right unshaded button at the top of the **Surfaces** selection list, and then select the desired surfaces from the **Surfaces** selection list.

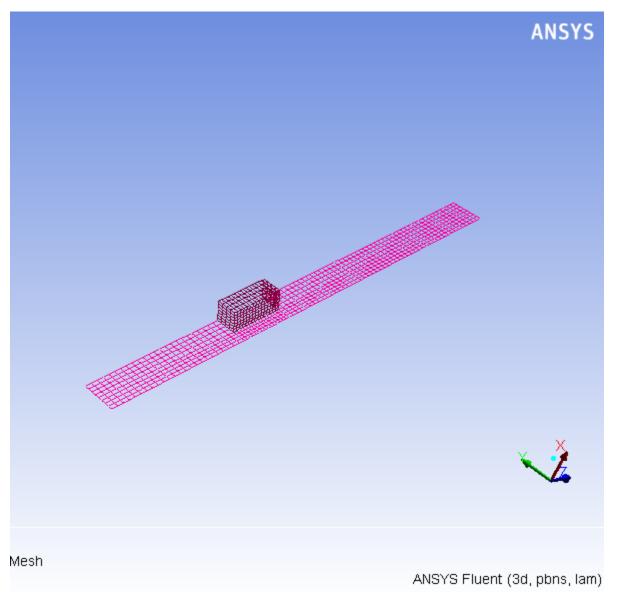
- c. Click the Colors... button to open the Mesh Colors dialog box.
 - i. Select **Color by ID** in the **Options** group box.
 - ii. Click **Reset Colors** to reset the mesh colors to the default settings and close the **Mesh Colors** dialog box.

d. Click **Display**.

2. Rotate and zoom the view.

Use the left mouse button to rotate the view. Use the middle mouse button to zoom the view until you obtain an enlarged display of the circuit board in the region of the chip, as shown in Figure 26.2: Mesh Display of the Chip and Board Surfaces (p. 1079).





Extra

You can click the right mouse button on one of the mesh boundaries displayed in the graphics window and its zone number, name, and type will be printed 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.

 $\mathbf{\mathbf{\dot{\mathbf{\nabla}}}} \mathbf{General} \rightarrow \mathbf{Display...}$

- a. Disable Edges and enable Faces in the Options group box.
- b. Click **Display** and close the **Mesh Display** dialog box.

The surfaces run together with no shading to separate the chip from the board.

26.4.4. Adding Lights

1. Add lighting effects.

\bigcirc Graphics and Animations \rightarrow Options...

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).

Display Options	—
Rendering	Graphics Window
Line Width 1	Active Window Close
Point Symbol (+) Animation Option Wireframe	Color Scheme Workbench
 Double Buffering Outer Face Culling Hidden Line Removal Hidden Surface Removal Removal Method 	Lighting Attributes Ights On Lighting Gouraud
Hardware Z-buffer 👻	Layout
Display Timeout Timeout in seconds	 ✓ Titles ✓ Axes ✓ Logo Color White ▼
	Colormap Colormap Alignment
Apply Info Lights	Close Help

- a. Disable **Double Buffering** in the **Rendering** group box.
- b. Enable Lights On in the Lighting Attributes group box.
- c. Select Gouraud from the Lighting drop-down list.

Flat is the most basic lighting whereas *Gouraud* gives better color gradiation.

d. Click Apply and close the Display Options dialog box.

Shading will be added to the surface mesh display (Figure 26.3: Graphics Display with Default Lighting (p. 1081)).

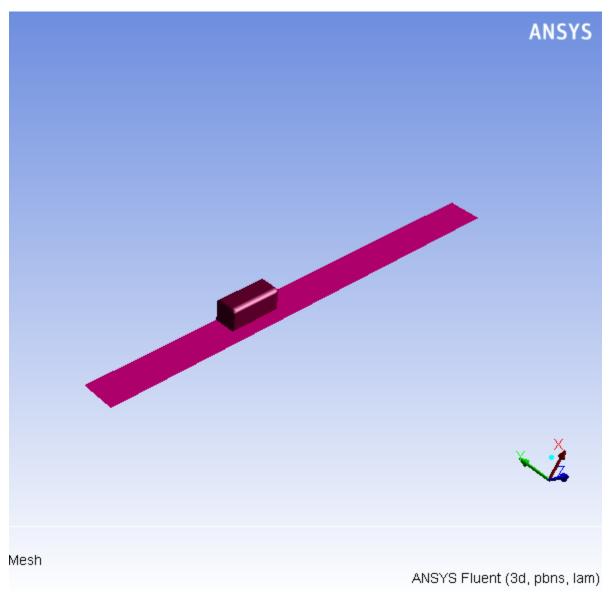


Figure 26.3: Graphics Display with Default Lighting

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. Enable Headlight On.

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 (Figure 26.4: Display with Additional Lighting: - Headlight Off(p. 1083)).

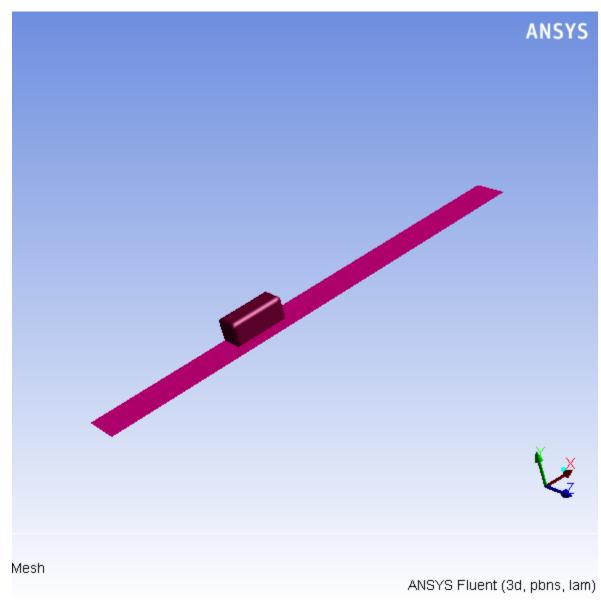
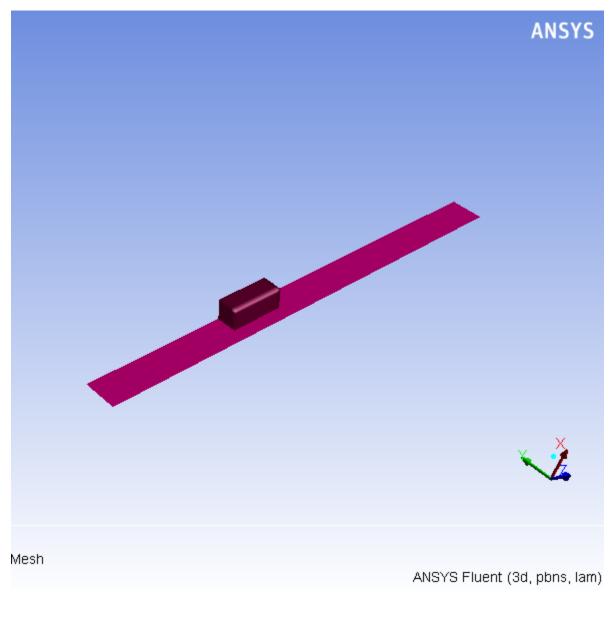


Figure 26.4: Display with Additional Lighting: - Headlight Off

- f. Click Apply.
- g. Similarly, add a second light (Light ID= 2) at (-1, 1, -1).

The result will be more softly shaded display (Figure 26.5: Display with Additional Lighting (p. 1084)).





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.

26.4.5. Creating Isosurfaces

To display results in a 3D model, you will need surfaces on which the data can be displayed. ANSYS 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, for example, 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.

1. Create a surface of constant *y* coordinate.

Surface \rightarrow Iso-Surface...

Iso-Surface		×
Surface of Constant Mesh Y-Coordinate Min (in) Max (in) 0 1.1 Iso-Values (in) 0.25 New Surface Name y=0.25in	solid-2	
Create Compute Manage	Close Help	

- 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.
- e. Click **Create** and close the **Iso-Surface** dialog box.
- 2. Create a clipped surface for the X-coordinate of the fluid (fluid-sym).

Surface → Iso-Clip...

Iso-Clip	••••
Clip to Values of Mesh X-Coordinate Min (in) Max (in) 1.9 3.9 X-Coordinate	Clip Surface board-ends board-sym board-top bottom-wall chip chip-bottom chip-sym fluid-sym pressure-outlet-16 symmetry-13 symmetry-19 top-wall New Surface Name fluid-sym-x-clip
Clip Compute Manage	Close Help

- 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.
- 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).

Surface \rightarrow Iso-Clip...

💶 Iso-Clip	
Mesh Y-Coordinate Y-Coordinate	Clip Surface board-top bottom-wall chip chip-bottom chip-sym fluid-sym-x-clip pressure-outlet-16 symmetry-13 symmetry-19 top-wall velocity-inlet-17 New Surface Name fluid-sym-y-clip
Clip Compute 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.

Note

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.

26.4.6. Generating Contours

1. Display filled contours of temperature on the symmetry plane (Figure 26.6: Filled Contours of Temperature on the Symmetry Surfaces (p. 1089)).

Contours		
Options	Contours of	
V Filled	Temperature 👻	
Vode Values	Static Temperature	
 Global Range Auto Range 	Min Max	
Clip to Range		
Draw Profiles		
Draw Mesh	Surfaces 🔋 🗏 🚍	
	chip-bottom 🔺	
Levels Setup	chip-sym fluid-sym	
20 🔺 1	fluid-sym-x-clip	
	fluid-sym-y-clip 👻	
Surface Name Pattern		
Match	New Surface 💌	
[Hater	Surface Types	
	axis	
	dip-surf exhaust-fan	
	fan v	
	[<u> </u>	
Display Compute Close Help		

- a. Retain the default settings in the **Options** group box.
- b. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- c. Select board-sym, chip-sym, and fluid-sym from the Surfaces selection list.
- d. Click **Display**.
- e. Rotate and zoom the display using the left and middle mouse buttons, respectively, to obtain the view as shown in Figure 26.6: Filled Contours of Temperature on the Symmetry Surfaces (p. 1089).

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 clicking the Views... button in Graphics and Animations task page and use the Default button to reset the view.
- Press the **Ctrl** + **L** to revert to a previous graphics display.

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.

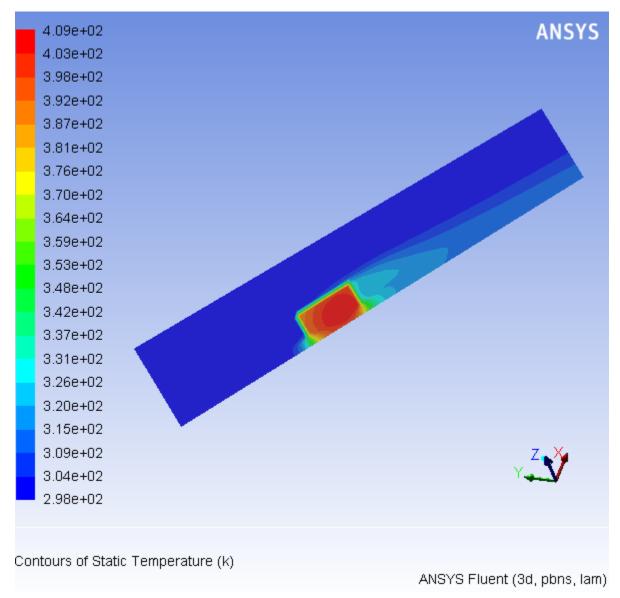


Figure 26.6: Filled Contours of Temperature on the Symmetry Surfaces

2. Display filled contours of temperature for the clipped surface (Figure 26.7: Filled Contours of Temperature on the Clipped Surface (p. 1090)).



- a. Deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym-x-clip** and **fluid-sym-y-clip**.
- b. Click **Display**.

A clipped surface appears, colored by temperature.

c. Orient the view to obtain the display as shown in Figure 26.7: Filled Contours of Temperature on the Clipped Surface (p. 1090).

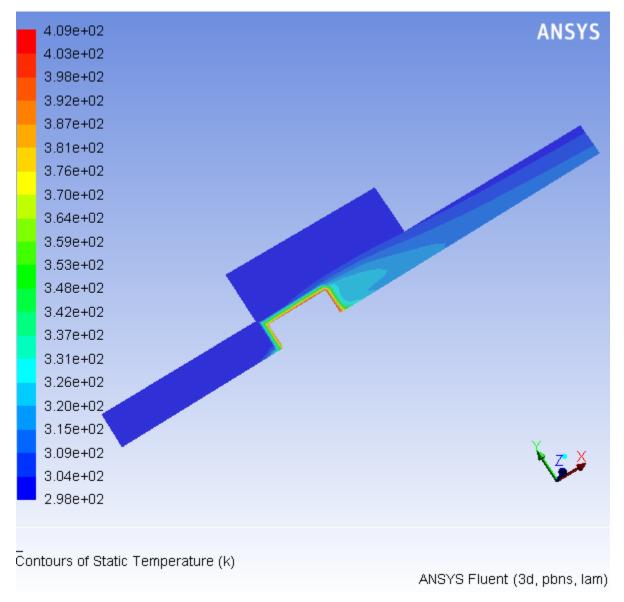


Figure 26.7: Filled Contours of Temperature on the Clipped Surface

3. Display filled contours of temperature on the plane, **y=0.25in** (Figure 26.8: Temperature Contours on the Surface, y = 0.25 in. (p. 1092)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

- a. Deselect all surfaces from the Surfaces selection list and then select y=0.25in.
- b. Click **Display** and close the **Contours** dialog box.

The filled temperature contours will be displayed on the **y=0.25in** plane.

4. Change the location of the colormap in the graphics display window.

• Graphics and Animations \rightarrow Options...

Display Options	
Rendering	Graphics Window
Line Width 1 Point Symbol (+) • Animation Option Wireframe • Double Buffering Outer Face Culling Hidden Line Removal Wirden Surface Removal Removal Method	Active Window Close 1 Set Color Scheme Workbench Lighting Attributes I Lighting Gouraud
Hardware Z-buffer	Layout
Display Timeout Timeout in seconds 60	 ✓ Titles Axes ✓ Logo Color White ▼ ✓ Colormap Colormap Alignment Bottom ▼
Apply Info Lights	Close Help

- a. Disable **Axes** in the **Layout** group box.
- b. Select Bottom from the Colormap Alignment drop-down list.
- c. Click Apply and close the Display Options dialog box.
- 5. Change the display of the colormap labels.

Graphics and Animations → Colormap...

Colormap	
Labels	Colormap
Show All Skip 4 Vumber Format Type exponential Precision 2	Log Scale Colormap Size 20 Currently Defined bgr
Apply Edit	Close Help

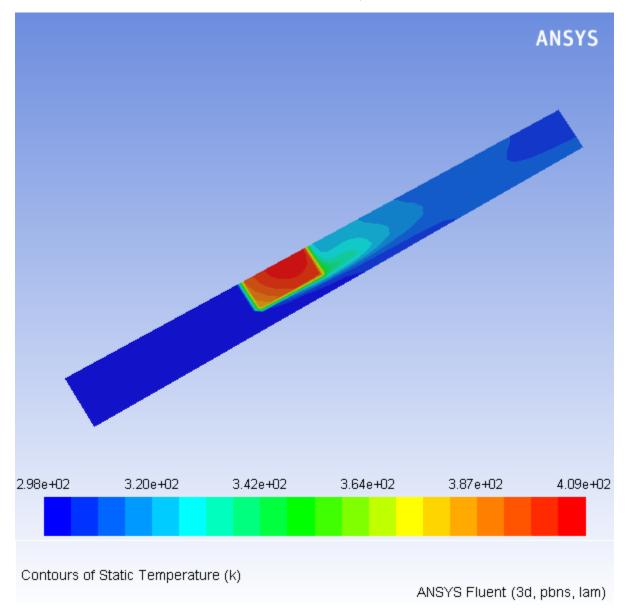
Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- a. Ensure that **Show All** is disabled.
- b. Set **Skip** to 4.

If the contour labels displayed next to the colormap are crowding the graphics window, you can use the skip-label function to control the number of labels displayed.

- c. Click Apply and close the Colormap dialog box.
- d. Zoom the display using the middle mouse button to obtain the view as shown in Figure 26.8: Temperature Contours on the Surface, y = 0.25 in. (p. 1092).

Figure 26.8: Temperature Contours on the Surface, y = 0.25 in.



In Figure 26.8: Temperature Contours on the Surface, y = 0.25 in. (p. 1092), 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.

26.4.7. 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 26.9: Velocity Vectors in the Module Symmetry Plane (p. 1095)).

E Vectors		×		
Options	Vectors of			
🔽 Global Range	Velocity	•		
Auto Range	Color by			
Clip to Range	Velocity	•		
Draw Mesh	Velocity Magnitude	•		
Style	Min (m/s) Max (m/s)			
headless	. 0.01532904 1.40682			
Scale Skip 1.9 0	Surfaces			
Vector Options	chip chip-bottom			
	chip-sym	E		
Custom Vectors	fluid-sym			
	fluid-sym-x-clip	-		
Surface Name Pattern	New Surface 🗸			
	Surface Types			
	axis			
	clip-surf exhaust-fan			
	fan	-		
	I			
Display Compute Close Help				

 \bigcirc Graphics and Animations $\rightarrow \equiv$ Vectors \rightarrow Set Up.

- a. Enter 1.9 for Scale.
- b. Deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym**.
- c. Click **Display** and close the **Vectors** dialog box.

Extra

You can display velocity vectors for the clipped surfaces.

Graphics and Animations → \blacksquare Vectors → Set Up...

a. Deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym-x-clip** and **fluid-sym-y-clip**.

- b. Click **Display**.
- 2. Change the colormap layout.

 $\clubsuit Graphics and Animations \rightarrow Options...$

- a. Enable **Axes** in the **Layout** group box.
- b. Select Left from the Colormap Alignment drop-down list.
- c. Click Apply and close the Display Options dialog box.
- d. Rotate and zoom the display to observe the vortex near the stagnation point and in the wake of the module (Figure 26.9: Velocity Vectors in the Module Symmetry Plane (p. 1095)).

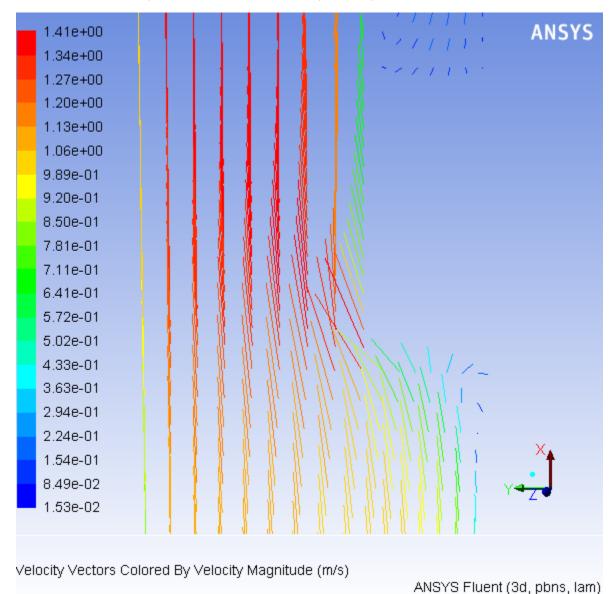


Figure 26.9: Velocity Vectors in the Module Symmetry Plane

Note

The vectors in Figure 26.9: Velocity Vectors in the Module Symmetry Plane (p. 1095) are shown without arrowheads. You can modify the arrow style in the **Vectors** dialog box by selecting a different option from the **Style** drop-down list.

Extra

If you want to decrease the number of vectors displayed, then increase the **Skip** factor to a non-zero value.

3. Plot velocity vectors in the horizontal plane intersecting the module (Figure 26.10: Velocity Vectors Intersecting the Surface (p. 1097)).

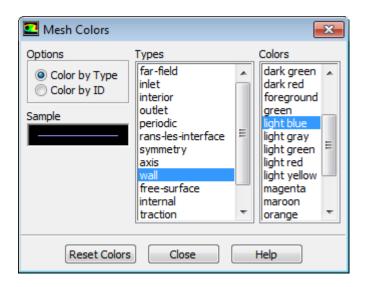
Graphics and Animations → \blacksquare Vectors → Set Up...

After plotting the vectors, you will enhance the view by mirroring the model about the module centerline and displaying the module surfaces.

a. Enable Draw Mesh in the Options group box to open the Mesh Display dialog box.

💶 Mesh Displa	у		— X—	
Options	Edge Type	Surfaces		
Nodes		board-sym		
Edges	Feature	board-top		
Faces	Outline	bottom-wall	=	
Partitions		chip		
		chip-bottom		
Shrink Factor	Feature Angle	chip-sym		
0	20	fluid-sym	-	
Surface Name Pa	attern Match	New Surface		
	Match	Surface Types		
Outline Inter	rior	axis		
		clip-surf		
Adjacency		exhaust-fan		
		fan	-	
Display Colors Close Help				

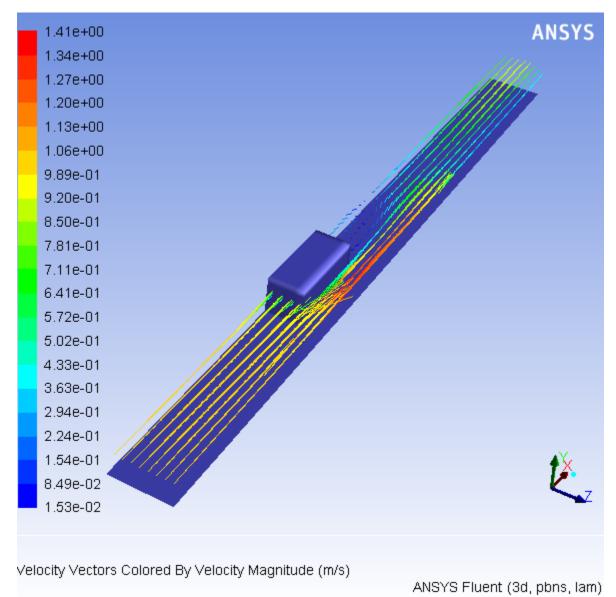
- i. Ensure that **Faces** is enabled in the **Options** group box.
- ii. Retain the selection of **board-top** and **chip** from the **Surfaces** selection list.
- iii. Click the Colors... button to open the MeshColors dialog box.



- 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 by clicking the unshaded icon to the right of the Surfaces selection list.
- d. Select y=0.25in from the Surfaces selection list.
- e. Click **Display** and close the **Vectors** dialog box.
- f. Rotate the model with the mouse to obtain the view as shown in Figure 26.10: Velocity Vectors Intersecting the Surface (p. 1097).

Figure 26.10: Velocity Vectors Intersecting the Surface



4. Mirror the image about the chip symmetry plane (Figure 26.11: Velocity Vectors After Mirroring (p. 1099)).

Graphics and Animations → Views...

D Views			
Views back bottom front isometric left right top Save Name view-0	Actions Default Auto Scale Previous Save Delete Read Write	Mirror Planes = symmetry-19 symmetry-18 symmetry-7 symmetry-12 symmetry-13 Define Plane Periodic Repeats Define	
Apply Camera Close Help			

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 26.11: Velocity Vectors After Mirroring (p. 1099)).

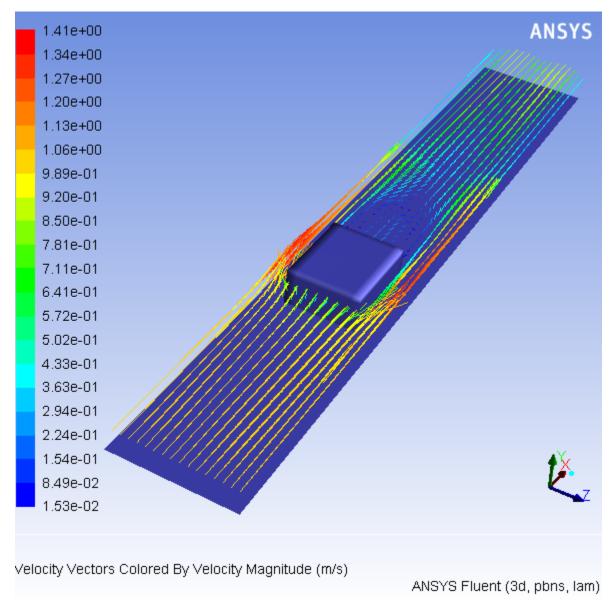


Figure 26.11: Velocity Vectors After Mirroring

26.4.8. Creating Animation

Using ANSYS Fluent, you can animate the solution and also a scene. For information on animating the solution, see Using Dynamic Meshes (p. 631), Step 10. In this tutorial, you will animate a scene between two static views of the graphics display.

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 26.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 1101)).



Contours			
Options	Contours of		
V Filled	Temperature 👻		
Vode Values Global Range	Static Temperature 🗸		
Auto Range	Min (k) Max (k)		
Clip to Range	297.9999 408.8104		
Draw Mesh	Surfaces		
	board-sym		
Levels Setup	board-top		
	bottom-wall		
20 🔺 1 🛋			
	chip-bottom		
Surface Name Pattern	New Surface 🔻		
Match	Surface Types 🔋 🗏 🗐		
	axis		
	clip-surf		
	exhaust-fan		
	fan 👻		
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 by clicking the unshaded icon to the right of **Surfaces**.
- d. Select **board-top** and **chip** from the **Surfaces** selection list.
- e. Click **Display** and close the **Contours** dialog box.
- f. Zoom the display as needed to obtain the view shown in Figure 26.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 1101).

Figure 26.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 1101) shows the high temperatures on the downstream portions of the module and relatively localized heating of the circuit board around the module.

4.09e+02	ANSYS
4.03e+02	
3.98e+02	
3.92e+02	
3.87e+02	
3.81e+02	
3.76e+02	
3.70e+02	
3.64e+02	
3.59e+02	
3.53e+02	
3.48e+02	
3.42e+02	
3.37e+02	
3.31e+02	
3.26e+02	
3.20e+02	
3.15e+02	
3.09e+02	
3.04e+02	
2.98e+02	
antours of Static Temperature (14)	
ontours of Static Temperature (k)	ANSYS Fluent (3d, pbns, larr

Figure 26.12: Filled Temperature Contours on the Chip and Board Top Surfaces

2. Create the key frames by changing the point of view.

Graphics and Animations → $\overline{\Xi}$ Scene Animation → Set Up...

💶 Animate	×
Playback Playback Mode Play Once Start Frame Increment End Frame 1 1 1 1 1 1 1 1 1 1 1 1 1	Key Frames Frame Keys Add Delete Delete All
Write/Record Format Key Frames	Picture Options
Write Read Clo	Belp

You will use the current display (Figure 26.12: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 1101)) 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. Zoom 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.

💶 Animate	
Playback Playback Mode Play Once Start Frame Increment End Frame 1 1 1 10 1 10 10 10 Frame	Key Frames Frame 10 Add Delete Delete All
Write/Record Format Key Frames	Picture Options
Write Read Clo	ise Help

The zoomed 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 zoom out the display so that the downstream side of the module is in the foreground (Figure 26.13: Filled Temperature Contours on the Chip and Board Top Surfaces (p. 1103)).

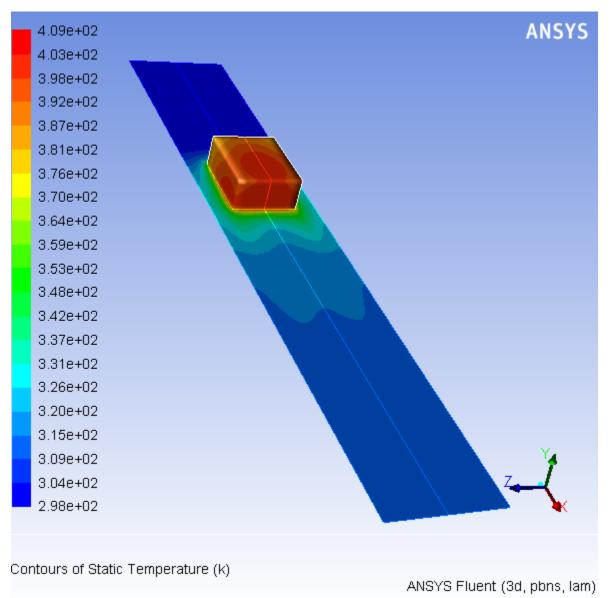


Figure 26.13: Filled Temperature Contours on the Chip and Board Top Surfaces

- 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 on the "play" button (

Note

If your computer is having difficulty displaying the animation with 200 frames, you can increase the increment to 10.

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 ANSYS Fluent creates between the key frames and thus stretching out your animation.

Note

You can also make use of animation tools of ANSYS Fluent for transient cases as demonstrated in Modeling Transient Compressible Flow (p. 257).

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 **Playback** modes, you can click the "stop"

button ()) to stop the continuous animation.

4. Close the **Animate** dialog box.

26.4.9. 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.

Surface → Line/Rake...

Iine/Rake Surface	
Options Type Line Tool Reset	Number of Points
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 26.14: Pathlines Display (p. 1107)).

♀Graphics and Animations → **\equiv** Pathlines → Set Up...

Pathlines						- X-
Options	Style			Color by		
Oil Flow	line		•	Particle Variables		•
Reverse Node Values	Att	ributes		Particle ID		•
Auto Range	Step Size (in)	olerance		Min	Max	
Draw Mesh	0.001	0.001		0	0	
Accuracy Control Relative Pathlines	Steps	Path Skip		Release from Surfaces		
XY Plot	6000	0		pathline-rake		*
Type CFD-Post	Path Coarsen			pressure-outlet-16 symmetry-13 symmetry-19		
Pulse Mode	On Zone	16		top-wall velocity-inlet-17 v=0.25in		=
Continuous Single	symmetry-12 symmetry-13 symmetry-18 symmetry-19 symmetry-7		-	Highlight Surfaces		•
Display	Pulse Co	ompute Axe	s	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, Step 6: Velocity Vectors.

- iii. Close the Mesh Display dialog box.
- b. Enter 0.001 inch for Step Size.
- c. Enter 6000 for **Steps**.

Note

A simple rule of thumb 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 the model so that the flow field is in front and the wake of the chip is visible as shown in Figure 26.14: Pathlines Display (p. 1107).

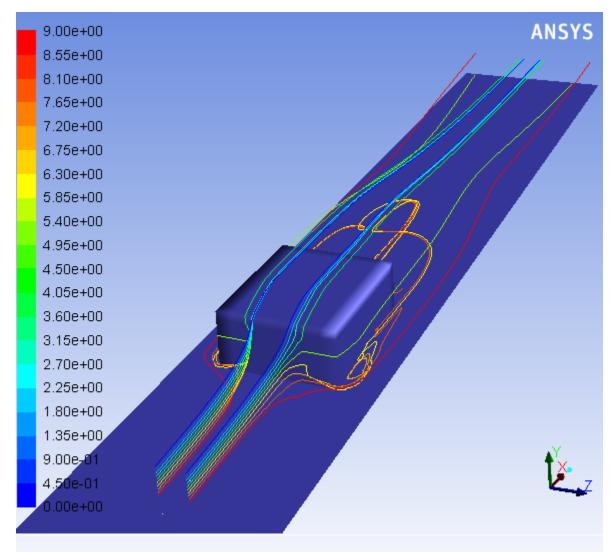


Figure 26.14: Pathlines Display

Pathlines Colored by Particle ID

ANSYS Fluent (3d, pbns, lam)

- 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.

ANSYS Fluent will save the file in Fieldview format.fvp extension.

4. Display pathlines as spheres.

Graphics and Animations → \blacksquare Pathlines → Set Up...

Pathlines		×
Options	Style	Color by
Oil Flow	sphere 🗸	Particle Variables 🔻
Reverse Vode Values	Attributes	Particle ID 🗸
Auto Range Draw Mesh Accuracy Control	Step Size (in) Tolerance 1 0.001	Min Max O 9
Relative Pathlines XY Plot Write to File	Steps Path Skip	
Type Fieldview	Path Coarsen	pressure-outlet-16 symmetry-13 symmetry-19 top-wall velocity-inlet-17
Pulse Mode Continuous Single	pressure-outlet-16 symmetry-12 symmetry-13 symmetry-18 symmetry-19 symmetry-7	velocity inter17
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

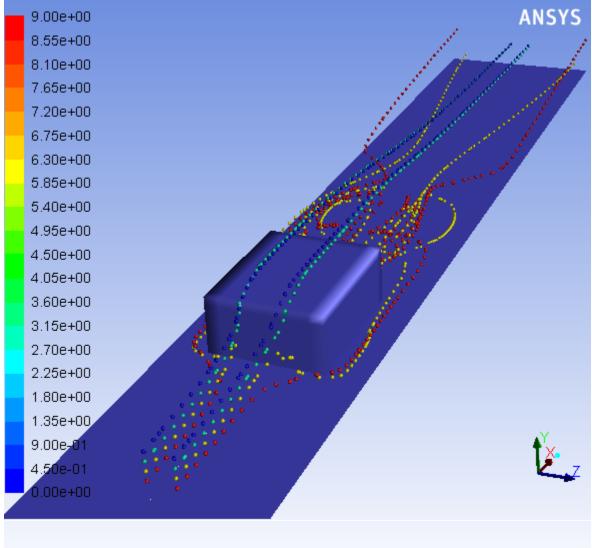
i. Enter 0.0005 for Diameter.

- 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.

h. Rotate the model so that the flow field is in front and the wake of the chip is visible as shown in Figure 26.15: Sphere Pathlines Display (p. 1109).

Figure 26.15: Sphere Pathlines Display



Pathlines Colored by Particle ID

ANSYS Fluent 15.0 (3d, pbns, lam)

i. Select **Surface ID** from the lower **Color by** drop-down list.

j. Click **Display** and close the **Pathlines** dialog box.

This will color the pathlines by the surface they are released from (Figure 26.16: Sphere Pathlines Colored by Surface ID (p. 1110)).

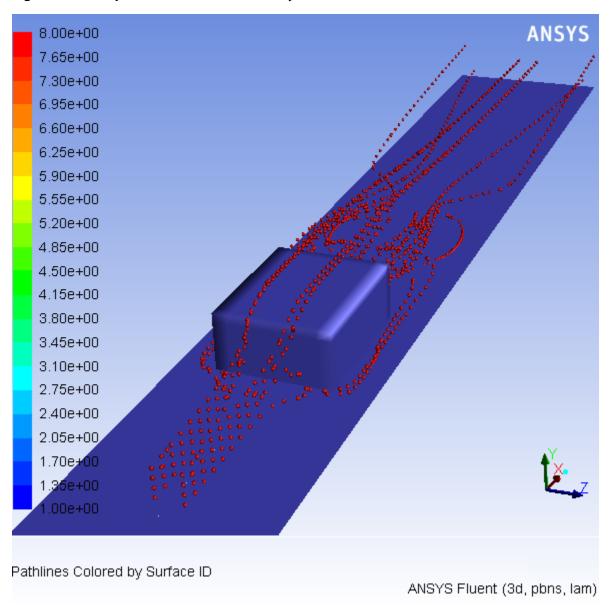


Figure 26.16: Sphere Pathlines Colored by Surface ID

Note

As an optional exercise, you can create solution animations for pathlines using **Animation Sequence** dialog box.

Calculation Activities (Solution Animations) \rightarrow Create/Edit...

26.4.10. Overlaying Velocity Vectors on the Pathline Display

The overlay capability, provided in the **Scene Description** dialog box, allows you to display multiple results on a single plot. You can exercise this capability by adding a velocity vector display to the pathlines just plotted.

1. Enable the overlays feature.

 $\clubsuit Graphics and Animations \rightarrow Scene...$

Scene Descriptio	on	×
Names (board-top chip path-8-surface-id	Geometry Attributes Type No geometry Display Transform Iso-Value Pathlines Ty	Scene Composition Overlays Draw Frame Frame Options
	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 vectors on the chip centerline plane.



Vectors			-X
Options	Vectors of		
Global Range	Velocity		•
Auto Range	Color by		
Clip to Range	Velocity		-
V Auto Scale			
Draw Mesh	Velocity Magnitude		•
Style	Min (m/s)	Max (m/s)	
headless 🗸	0.01532904	1.40682	
Scale Skip	Surfaces		
3.8 0	bottom-wall		
	chip		
Vector Options	chip-bottom		E
Custom Vectors	chip-sym		
()	fluid-sym		
	fluid-sym-x-clip		-
Surface Name Pattern Match	New Surface 🔻		
	Surface Types		
	axis		
	clip-surf		
	exhaust-fan		
	fan		-
Display	Compute Close	Help	

- a. Disable Draw Mesh in the Options group box.
- b. Retain the value of 3.8 for **Scale**.
- c. Deselect all surfaces by clicking the unshaded icon to the right of Surfaces.
- d. Select fluid-sym from the Surfaces selection list.

Because the mesh surfaces are already displayed and overlaying is active, there is no need to redisplay the mesh surfaces.

- e. Click **Display** and close the **Vectors** dialog box.
- f. Use the mouse to obtain the view that is shown in Figure 26.17: Overlay of Velocity Vectors and Pathlines Display (p. 1113).

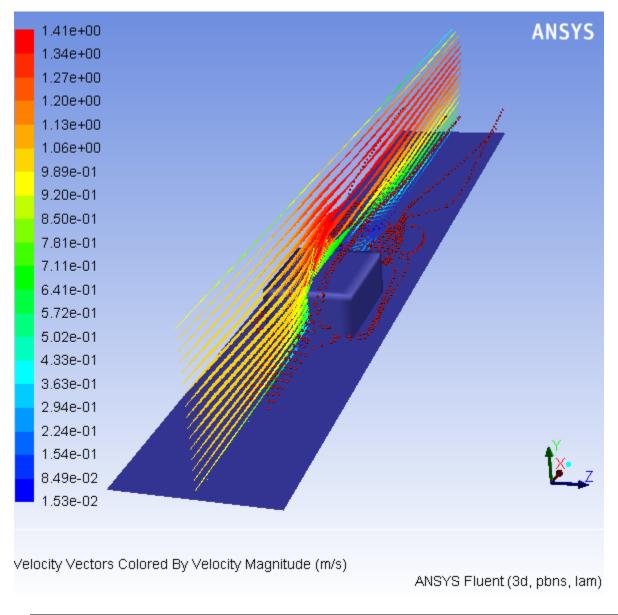


Figure 26.17: Overlay of Velocity Vectors and Pathlines Display

Note

The final display (Figure 26.17: Overlay of Velocity Vectors and Pathlines Display (p. 1113)) does not require mirroring about the symmetry plane because the vectors obscure the mirrored image. You may disable the mirroring option in the **Views** dialog box at any stage during this exercise.

26.4.11. 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. Delete the velocity vectors and pathlines from the current display.

Graphics and Animations → Scene...

Scene Descriptio	n		—
Names	4	Geometry Attributes Type Group Display Transform Iso-Value Pathlines Time Step	Scene Composition Overlays Draw Frame Frame Options
	Ap	ply Close Help	

- a. Select the pathlines **path-8-surface-id** and the velocity vectors **vv-0-velocity-magnitude** from the **Names** selection list.
- b. Click **Delete Geometry**.
- c. Click Apply and close the Scene Description dialog box.

The **Names** selection list of the **Scene Description** dialog box should then contain only the two mesh surfaces (**board-top** and **chip**).

2. Create a plotting surface at *x*=3 inches (named **x=3.0in**), just downstream of the trailing edge of the module.

Surface \rightarrow Iso-Surface...

Iso-Surface	
Surface of Constant	From Surface
Mesh 👻	board-ends
X-Coordinate	board-sym
Min Max	board-top bottom-wall
	chip
	chip-bottom 🚽
Iso-Values	
3.0	From Zones
<>	fluid-8 solid-1
New Surface Name	solid-2
x=3.0in	
Create Compute Manage.	Close Help

Tip

For details on creating an isosurface, see Step 4: Creating Isosurfaces.

3. Add the display of filled temperature contours on the **x=3.0in** surface.

Contours	
Options Image: Clip to Range Image: Clip to Range	Contours of Temperature Static Temperature Min Max
Draw Profiles	0 0 Surfaces 3 3
Levels Setup	symmetry-19 top-wall velocity-inlet-17 x=3.0in y=0.25in
Surface Name Pattern	New Surface ▼ Surface Types
	axis dip-surf exhaust-fan fan -
Display	Compute Close Help

Graphics and Animations → $\stackrel{\frown}{\equiv}$ Contours → Set Up...

- a. Ensure that **Draw Mesh** is disabled in the **Options** group box.
- b. Deselect all surfaces by clicking on the unshaded icon to the right of **Surfaces**.
- 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.

-			
Vectors			×
Options	Vectors of		
Global Range	Velocity		•
V Auto Range	Color by		
Clip to Range	Velocity		•
Auto Scale Draw Mesh			
Uraw Mest	Velocity Magnitude		•
Style	Min	Max	
headless 🔹	0	0	
Scale Skip	Surfaces		
1.9 2	symmetry-13		
	symmetry-19		
Vector Options	top-wall		
Custom Vectors	velocity-inlet-17		
Custoin vectors	x=3.0in		E
	y=0.25in		-
Surface Name Pattern			
Match	New Surface 🔻		
	Surface Types		
	axis		*
	clip-surf		
	exhaust-fan		
	fan -		Ψ.
Display	Compute Close	Help	

• Graphics and Animations $\rightarrow \equiv$ Vectors \rightarrow Set Up...

- 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 by clicking on the unshaded icon to the right of 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 26.18: Exploded Scene Display of Temperature and Velocity (p. 1118)).

• Graphics and Animations \rightarrow Scene...

- a. Select contour-9-temperature from the Names selection list.
- b. Click the Transform... button to open the Transformations dialog box.

Transformatio	ns		x
Geometry Name contour-9-temper	ature	Meridional	
Translate	Rotate by	Rotate about	Scale
X (in)	X (deg)	X (in)	X 1
Y (in)	Y (deg)	Y (in)	Y 1
Z (in)	Z (deg) 0	Z (in)	Z 1
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 26.18: Exploded Scene Display of Temperature and Velocity (p. 1118)).

- c. Deselect **Overlays**.
- d. Click Apply and close the Scene Description dialog box.

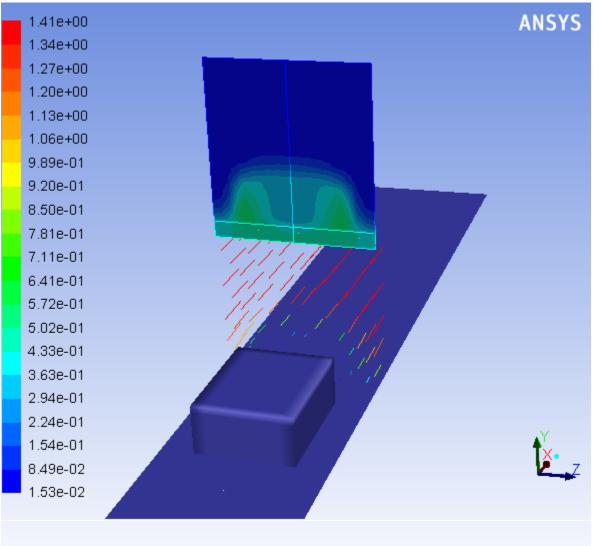


Figure 26.18: Exploded Scene Display of Temperature and Velocity

Velocity Vectors Colored By Velocity Magnitude (m/s)

ANSYS Fluent (3d, pbns, lam)

26.4.12. 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.

Graphics and Animations → Scene...

- a. Select contour-9-temperature and vv-9-velocity-magnitude from the Names selection list.
- b. Click Delete Geometry.
- c. Click **Apply** and close the **Scene Description** dialog box.

The dialog box and display window will be updated to contain only the mesh surfaces.

- 2. Use the mouse to zoom out the view in the graphics window so that the entire board surface is visible.
- 3. Generate contours of velocity magnitude and sweep them through the domain along the X axis.

Sweep Surface	1	×
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	Value 0.08255	Max Value 0.1651
•		÷.
Create	Animate Compute	Close Help

\bigcirc Graphics and Animations → **\equiv** Sweep Surface → Set Up...

- 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 in the Display Type list to open the Contours dialog box.

Contours	
Options	Contours of
V Filled	Velocity 👻
Vode Values	Velocity Magnitude
 Global Range Auto Range 	Min Max
Clip to Range	0
Draw Profiles	
Draw Mesh	Surfaces 🔋 🗏 🗏
	symmetry-19
Levels Setup	top-wall
	velocity-inlet-17
20	x=3.0in
	y=0.25in 👻
Surface Name Pattern	New Surface -
Match	Surface Types
	axis
	clip-surf
	exhaust-fan
	fan 👻
ОК	Compute Cancel Help

- i. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- ii. 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. ANSYS 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 ANSYS Fluent for transient cases as demonstrated in Modeling Transient Compressible Flow (p. 257).

26.4.13. 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.

```
Surface \rightarrow Line/Rake...
```

Line/Rake Surface	
Options Type Line Tool Reset	Number of Points
End Points	
x0 (in) 2	x1 (in) 2.75
y0 (in) 0.4	y1 (in) 0.4
z0 (in) 0.01	z1 (in) 0.01
Select Poin	ts 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 26.19: Temperature Along the Top Centerline of the Module (p. 1123)).

♦ Plots $\rightarrow \stackrel{\bullet}{=}$ XY Plot \rightarrow Set Up...

Solution XY Plot			-X
Options	Plot Direction	Y Axis Function	
Vode Values	X 1	Temperature	•
Position on X Axis Position on Y Axis	Y	Static Temperature	•
Write to File	Y O	X Axis Function	
Order Points	Z O	Direction Vector	
File Data 🔳 🔳 🚍	j 🗀 🔤	Surfaces	
]	pressure-outlet-16	*
		symmetry-13 symmetry-19	
		top-center-line	
		top-wall velocity-inlet-17	=
		x=3.0in	_
	Load File	L.o. 25-	•
	Free Data	New Surface 🕶	
Plot	Axes	Curves Close Help	

- a. Retain the default **Plot Direction** of **X**.
- 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**).

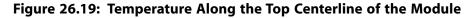
d. Click the Axes... button to open the Axes - Solution XY Plot dialog box.

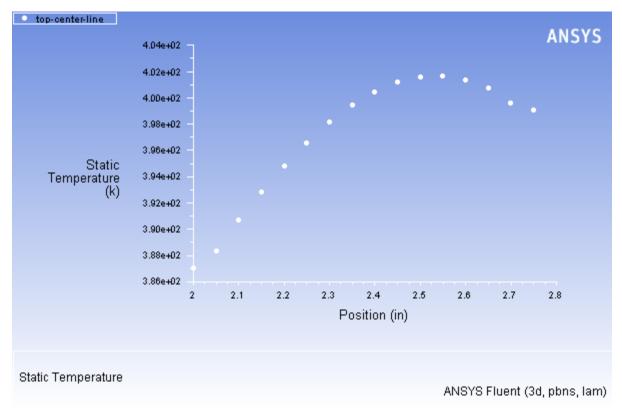
Axes - Solution XY Plot		—
Axis X Y Label	Number Format Type general Precision 3	Major Rules Color foreground Weight 1
Options Log Auto Range Major Rules Minor Rules	Range Minimum 2.0 Maximum 2.75	Minor Rules
	Apply Close He	lp

- 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 **Plot** and close the **Solution XY Plot** dialog box.

The temperature distribution (Figure 26.19: Temperature Along the Top Centerline of the Module (p. 1123)) shows the temperature increase across the module surface as the thermal boundary layer develops in the cooling air flow.





26.4.14. Creating Annotation

Graphics and Animations → Annotate...

You can annotate the display with the text of your choice.

Annotate		— ×
Names 🕃 🚍	Annotation Text Temperature Along the Top Centerline Font Specification Name Sans Serif Color Slant foreground Regular	Weight Medium Size 20
	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 display window where you want the text to appear, and you will see the text displayed at the selected location (Figure 26.20: Temperature Along the Top Centerline of the Module (p. 1125)).

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 hardcopy 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 hardcopy 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 hardcopy.

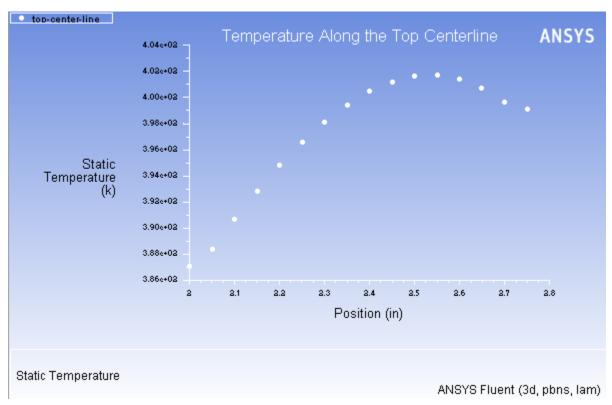


Figure 26.20: Temperature Along the Top Centerline of the Module

26.4.15. Saving Hardcopy Files

File → Save Picture...

You can save hardcopy files of the graphics display 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.

Save Picture			—
© EPS © JPEG © PPM @ PostScript © TIFF	Coloring © Color © Gray Scale © Monochrome Options	File Type Raster Vector	Resolution DPI 75 A Height 720 P
PNG VRML Window Dump Save	Landscape Orier	itauon	dow Dump Command port -window %w Be Help

- 1. Select **PostScript** in the **Format** group box.
- 2. Select **Color** in the **Coloring** group box.
- 3. Select **Vector** in the **File Type** group box.

- 4. 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.
- 5. Close the Save Picture dialog box.

26.4.16. Generating Volume Integral Reports

$\clubsuit Reports \rightarrow \blacksquare Volume Integrals \rightarrow Set Up...$

Reports of Volume Integral can be used to determine the Volume of a particular fluid region (that is, 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.

Volume Integrals		×
Report Type Mass-Average Mass Integral Sum Minimum Minimum Maximum Volume Volume Volume Integral		Cell Zones 🕃 🔳 = fluid-8 solid-1 solid-2
	Compute Write Close Help	

- 1. Select Maximum in the Report Type group box.
- 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.

26.5. Summary

This tutorial demonstrated the use of many of the extensive postprocessing features available in ANSYS Fluent.

For more information on these and related features, see Reporting Alphanumeric Data or Displaying Graphics in User's Guide.

Chapter 27: Parallel Processing

This tutorial is divided into the following sections:

- 27.1. Introduction27.2. Prerequisites27.3. Problem Description27.4. Setup and Solution
- 27.5. Summary

27.1. Introduction

This tutorial illustrates the setup and solution of a simple 3D problem using 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.

The tutorial assumes that both ANSYS Fluent and network communication software have been correctly installed (see the separate installation instructions and related information for details). The case chosen is the mixing elbow problem you solved in Tutorial 3.

This tutorial demonstrates how to do the following:

- Start the parallel version of ANSYS Fluent using either Windows or Linux.
- · Partition a mesh for parallel processing.
- Use a parallel network of workstations.
- Check the performance of the parallel solver.

27.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

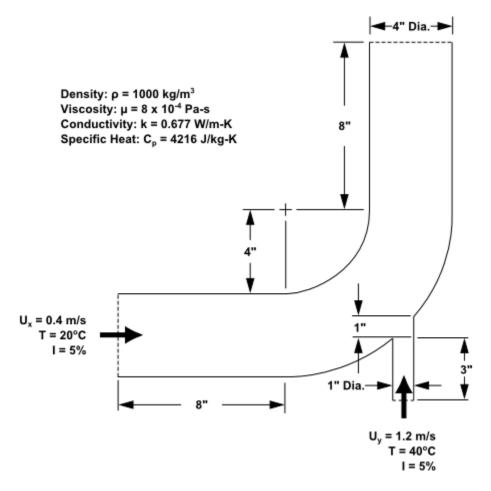
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

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

27.3. Problem Description

The problem to be considered is shown schematically in Figure 27.1: Problem Specification (p. 1130). A cold fluid at 20 $\degree C$ flows into the pipe through a large inlet, and mixes with a warmer fluid at 40 $\degree C$ 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.

Figure 27.1: Problem Specification



27.4. Setup and Solution

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

- 27.4.1. Preparation
- 27.4.2. Starting the Parallel Version of ANSYS Fluent
- 27.4.3. Reading and Partitioning the Mesh
- 27.4.4. Solution
- 27.4.5. Checking Parallel Performance
- 27.4.6. Postprocessing

27.4.1. 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 **15.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click Files to download the input and solution files.
- 7. Unzip parallel_process_R150.zip to your working folder.

The case file elbow2.cas.gz can be found in the parallel_process directory created after unzipping the file.

You can partition the mesh before or after you set up the problem (define models, boundary conditions, and so on.). It is best to partition after the problem is set up, since partitioning has some model dependencies (for example, sliding-mesh and shell-conduction encapsulation). Since you have already followed the procedure for setting up the mixing elbow in Tutorial 3, elbow2.cas.gz is provided to save you the effort of redefining the models and boundary conditions.

27.4.2. 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.

27.4.2.1. 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. Specify **3D** for **Dimension**.
- 2. Select Parallel (Local Machine) under Processing Options.
- 3. Set Number of 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**.

4. Click **OK**.

E Fluent Launcher	
ANSYS	Fluent Launcher
Dimension 2D 3D Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme	Options Double Precision Meshing Mode Use Job Scheduler Use Remote Linux Nodes Processing Options Serial Pranlel (Local Machine) Number of Processes
Show Fewer Options General Options Parallel Settings	2 🖨
Interconnects default MPI Types default Run Types	 Validate Platform MPI Password
 Shared Memory on Local Machin Distributed Memory on a Cluster 	e
<u>Q</u> K <u>D</u> efa	ault <u>C</u> ancel <u>H</u> elp v

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 Parallel Processing in the User's Guide.

27.4.2.2. 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. Specify **3D** for **Dimension**.
- 3. Select Parallel (Local Machine) under Processing Options.
- 4. Set the Number of Processes to 2.
- 5. Click the **Show More Options** button and select the **Parallel Settings** tab.
 - 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.

- Select Distributed Memory on a Cluster.
- Ensure that File Containing Machine Names is selected to specify the file.
- 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.

6. 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**.

I Fluent Launcher	
ANSYS	Fluent Launcher
Dimension 2D 3D Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme	Options Double Precision Meshing Mode Use Job Scheduler Use Remote Linux Nodes Processing Options Serial Prallel (Local Machine) Number of Processes
 Show Fewer Options General Options Parallel Settings Interconnects default MPI Types default 	Scheduler Environment
Run Types Shared Memory on Local Machin Distributed Memory on a Cluster Machine Names File Containing Machine Nam W:\doc\fluent140\fluent.hosts	
<u> </u>	ult <u>C</u> ancel <u>H</u> elp –

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:

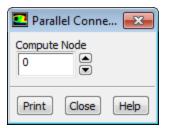
my_computer
another_computer

For additional information about hosts files and parallel command line options, see Parallel Processing in the User's Guide.

7. 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.

$\textbf{Parallel} \rightarrow \textbf{Network} \rightarrow \textbf{Show Connectivity...}$



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	Comm.	Hostname	0.S.	PID	Mach ID	HW ID	Name
host n1 n0*	net pcmpi pcmpi	my_computer another_computer my_computer	Windows-x64 Windows-x64 Windows-x64	2892	0 0 0	620 1 0	Fluent Host Fluent Node Fluent Node
Selected system interconnect: default							

ID is the sequential denomination of each compute node (the host process is always host), Comm. is the communication library (that is, MPI type), Hostname is the name of the machine hosting the compute node (or the host process), O.S. is the architecture, PID is the process ID number, Mach ID is the compute node ID, and HW ID is an identifier specific to the communicator used.

c. Close the Parallel Connectivity dialog box.

27.4.3. 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 → Auto Partition...

Auto Partition	Mesh	x
Method Metis V Case File		Optimizations Pre-Test
Across Zones		
	OK Cancel Help	

If the **Case File** option is selected (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. Retain the **Case File** option.

When the **Case File** option is selected, 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.

$\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Case...}$

3. Examine the front view of the **symmetry** mesh zone (Figure 27.2: Mesh Along the Symmetry Plane for the Mixing Elbow (p. 1137)).

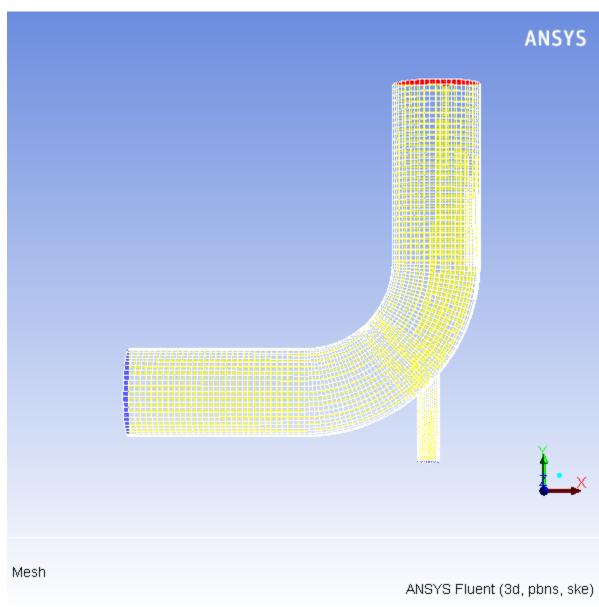


Figure 27.2: Mesh Along the Symmetry Plane for the Mixing Elbow

4. Check the partition information.

 $\textbf{Parallel} \rightarrow \textbf{Partitioning and Load Balancing...}$

Partitioning and Load Balancing	—
Method	
Metis	
Options Optimization Weighting Dynamic Load Balancing	Zones 🔋 🗏 🚍
Number of Partitions 2	fluid
Reporting Verbosity 1	
Across Zones	Registers 🖻 🗐 🚍
Reordering Methods	Registers 🗉 🖃 🖃
Architecture Aware Reverse Cuthill-McKee	
Curim-vickee	
Print Active Partitions Print Stored Partitions Use Stored Partitions	
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 Partitions:
           P Cells I-Cells Cell Ratio Faces I-Faces Face Ratio Neighbors Load
           0 11638 195 0.017 37996 263 0.007 1 1
1 11328 205 0.018 37565 263 0.007 1 1
                         _____
         Collective Partition Statistics:
                                                                                                                             Minimum Maximum Total
         _____
                                                                                                                             11328 11638
                                                                                                                                                                                            22966
         Cell count
                                                                                                                                  -1.3%
        -1.3% 1.3%

1.3% 1.3%

Partition boundary cell count ratio 1.7% 1.8%

Face the face 
         Mean cell count deviation
                                                                                                                                                                 1.3%
                                                                                                                                                                                             400
                                                                                                                                                             1.8%
                                                                                                                                                                                             1.7%
                                                                                                                                                       37996
0.6%
                                                                                                                               37565
        Face count
                                                                                                                                                                                                75298
         Mean face count deviation
                                                                                                                                  -0.6%
                                                                                                           263
         Partition boundary face count
                                                                                                                                                                 263
                                                                                                                                                                                                 263
         Partition boundary face count ratio 0.7%
                                                                                                                                                                                                 0.3%
                                                                                                                                                                0.7%
         Partition neighbor count
                                                                                                                                   1
                                                                                                                                                                  1
         _____
         Partition Method
                                                                                                                                   Metis
         Stored Partition Count
                                                                                                                                   2
Done.
```

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 **Partition Mesh** 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 **Partition Mesh** 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 Parallel Processing in the User's Guide.

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 bandwidth, and a minimum number of partition neighbors to reduce the startup time for communication. Here, 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.

♀Solution Initialization → Initialize

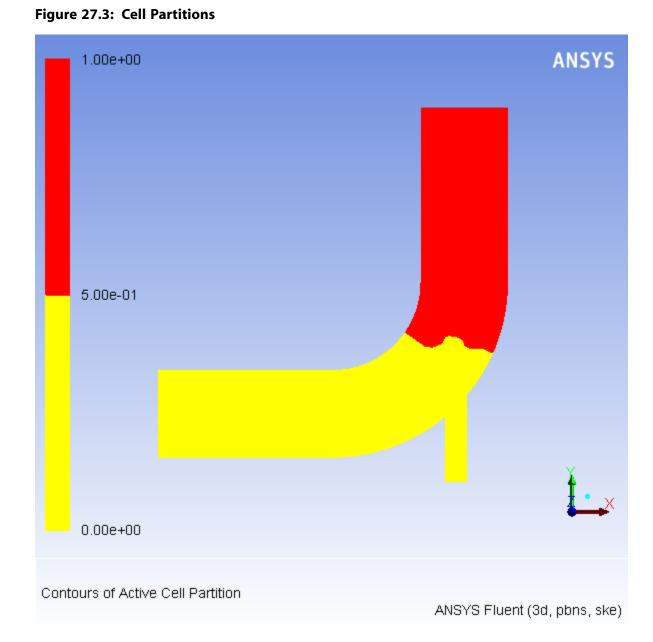
In order to use the **Contours** dialog box to inspect the partition you just created, you have to initialize the solution, even though you are not going to solve the problem at this point. The default values are sufficient for this initialization.

b. Display the cell partitions (Figure 27.3: Cell Partitions (p. 1141)).

Graphics and Animations → $\stackrel{\frown}{=}$ Contours → Set Up...

Contours	×
Options	Contours of
V Filled	Cell Info 👻
 Node Values Global Range 	Active Cell Partition 🗸
Auto Range	Min Max
Clip to Range	0 1
Draw Mesh	Surfaces
	default-interior
Levels Setup	pressure-outlet-7
2 1	velocity-inlet-5
	velocity-inlet-6
Surface Name Pattern	New Surface 🔻
Match	Surface Types
	axis
	dip-surf 📃
	fan -
	L
Display	Compute Close Help

- i. Ensure 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.



As shown in Figure 27.3: Cell Partitions (p. 1141), 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 dissatified with the partitions, you can use the **Partition Mesh** 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 **Partition Mesh** 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 Partitioning the Mesh Manually and Balancing the Load in the User's Guide.

6. Save the case file with the partitioned mesh (elbow3.cas.gz).

 $\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case...}$

27.4.4. Solution

1. Initialize the flow field.



Solution Initialization			
Initialization Methods			
 Hybrid Initialization Standard Initialization 			
More Settings			
Patch			
Reset DPM Sources Reset Statistics			
Help			

- a. Retain the default selection of Hybrid Initialization from the Initialization Methods group box.
- b. Click Initialize.

A **Warning** 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 **Warning** dialog box to discard the data.
- 2. Enable the plotting of residuals during the calculation.

```
    Monitors → 
    Eresiduals → Edit...
```

3. Start the calculation by requesting 90 iterations.

CRun Calculation

The solution will converge in approximately 36 iterations.

4. Save the data file (elbow3.dat.gz).

File \rightarrow Write \rightarrow Data...

27.4.5. 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 Parallel Processing in the User's Guide.

Parallel \rightarrow Timer \rightarrow Usage

Pe	erformance Timer for 36 iterations on 2 compute	nodes	
	Average wall-clock time per iteration:	0.107	sec
	Global reductions per iteration:	79	ops
	Global reductions time per iteration:	0.000	sec (0.0%)
	Message count per iteration:	1330	messages
	Data transfer per iteration:	0.470	MB
	LE solves per iteration:	4	solves
	LE wall-clock time per iteration:	0.058	sec (54.1%)
	LE global solves per iteration:	8	solves
	LE global wall-clock time per iteration:	0.000	sec (0.1%)
	LE global matrix maximum size:	17	
	AMG cycles per iteration:	8.139	cycles
	Relaxation sweeps per iteration:	853	sweeps
	Relaxation exchanges per iteration:	861	exchanges
	Total wall-clock time:	3.836	sec
	Total CPU time:	7.691	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. As a rough estimate of parallel performance, you can compare the Total wall-clock time with the Total CPU time. In this case, the CPU time was approximately twice the Total wall-clock time. For a parallel process run on two compute nodes, this reveals very good parallel performance, even though the advantage over a serial calculation is small, as expected for this simple 3D problem.

Note

The wall clock time, the CPU time, and the ratio of iterations to convergence time may differ depending on the type of computer you are running (for example, Windows 32, Linux 64, and so on).

27.4.6. Postprocessing

See Tutorial 3 for complete postprocessing exercises for this example. Here, two plots are generated so that you can confirm that the results obtained with the parallel solver are the same as those obtained with the serial solver.

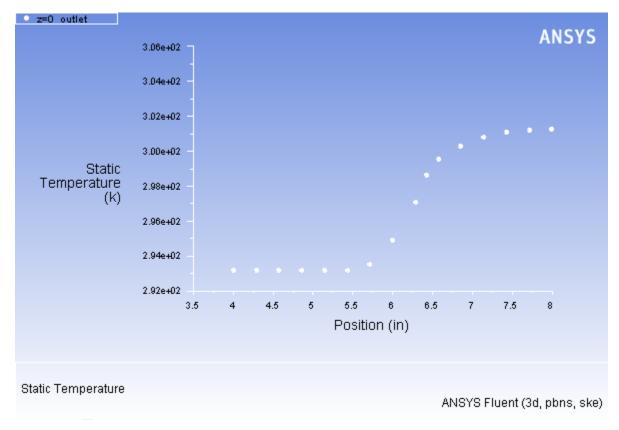
1. Display an XY plot of temperature along the centerline of the outlet (Figure 27.4: Temperature Distribution at the Outlet (p. 1144)).

$$\mathbf{O}_{\mathsf{Plots}} \to \mathbf{\overline{E}}_{\mathsf{XY}} \mathsf{Plot} \to \mathsf{Set} \mathsf{Up...}$$

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 Load File Free Data	Y Axis Function Temperature Static Temperature X Axis Function Direction Vector Surfaces Unifaces Default-interior pressure-outlet-7 symmetry 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 **z=0_outlet** from the **Surfaces** selection list.
- c. Click **Plot** and close the **Solution XY Plot** dialog box.

Figure 27.4: 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. 187).

2. Display filled contours of the custom field function **dynamic-head** (Figure 27.5: Contours of the Custom Field Function, Dynamic Head (p. 1146)).

Contours	
Options	Contours of
V Filled	Custom Field Functions 👻
 Node Values Global Range 	dynamic-head 🗸
Auto Range	Min Max
Clip to Range	0
Draw Profiles Draw Mesh	Surfaces
E bran rican	default-interior
Levels Setup	pressure-outlet-7
Levels Setup 80 1	symmetry
	velocity-inlet-5 velocity-inlet-6
Surface Name Pattern Match	New Surface 💌
Inatch	Surface Types
	axis
	clip-surf exhaust-fan
	fan 👻
	·- ·
Display	Compute Close Help

Graphics and Animations → \blacksquare Contours → Set Up...

a. Select **Custom Field Functions...** from the **Contours of** drop-down list.

The custom field function you created in Tutorial 3 (**dynamic-head**) will be selected in the lower drop-down list.

- b. Enter 80 for Levels.
- c. Select symmetry from the Surfaces selection list.
- d. Click **Display** and close the **Contours** dialog box.

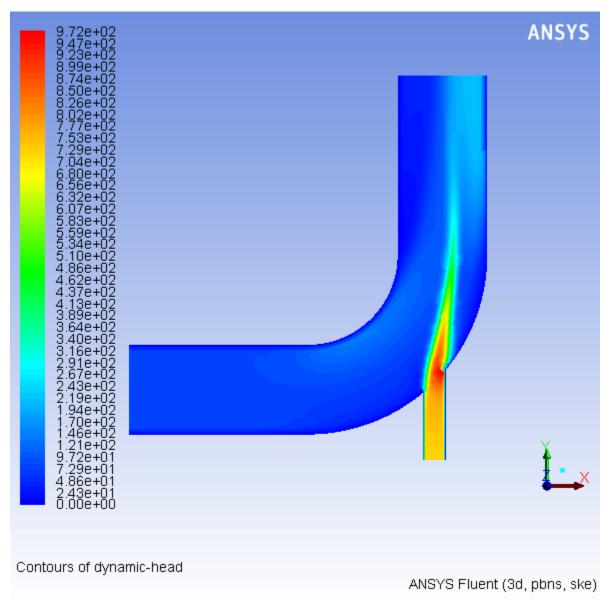


Figure 27.5: Contours of the Custom Field Function, Dynamic Head

27.5. Summary

This tutorial demonstrated how to solve a simple 3D problem using the parallel solver of ANSYS Fluent. Here, the automatic mesh partitioning performed by ANSYS Fluent when you read the mesh into the parallel version was found to be acceptable. You also learned how to check the performance of the parallel solver to determine if optimizations are required.

For additional details about using the parallel solver, see Checking and Improving Parallel Performance in the User's Guide.