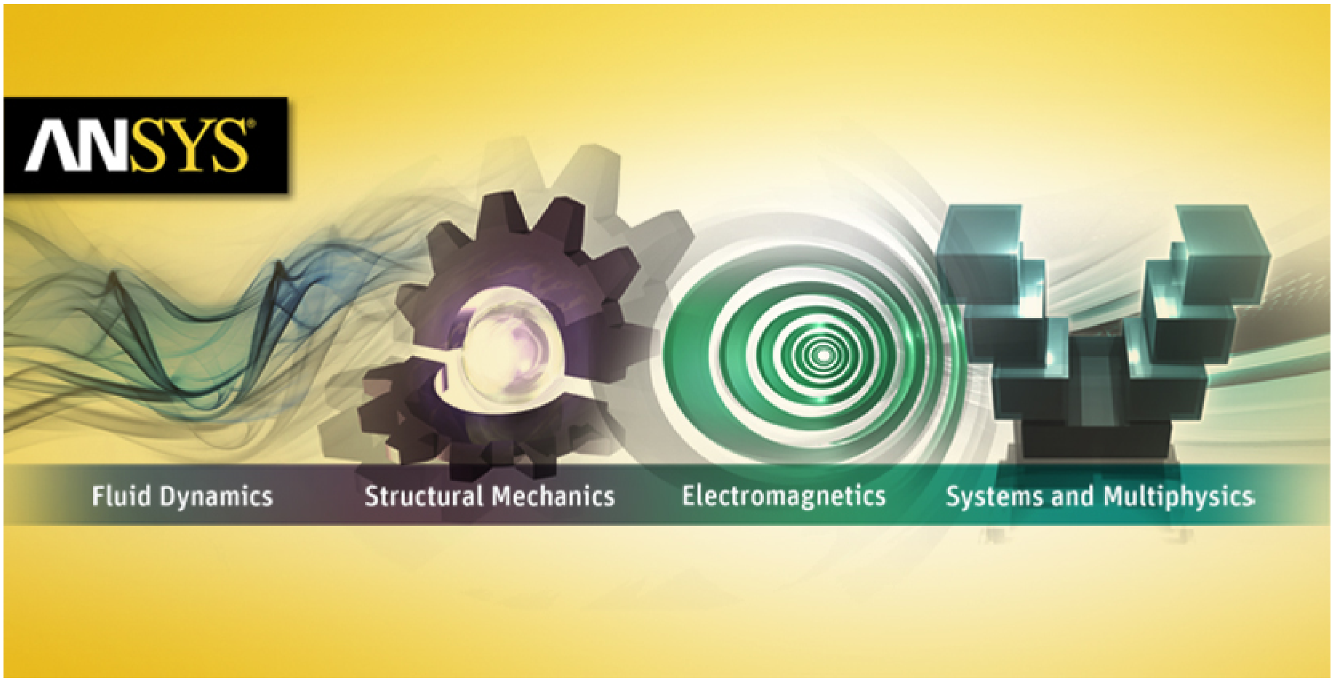


ANSYS Fluid Dynamics Verification Manual

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 14.0
August 2011

Published in the U.S.A.

Table of Contents

Verification Test Case Descriptions

Chapter 1: Introduction

The Verification Manual presents a collection of test cases that demonstrate a representative set of the capabilities of the **ANSYS Fluid Dynamics** product suite. The primary purpose of this manual is to demonstrate a wide range of capabilities in straightforward problems which have "classical" or readily-obtainable theoretical solutions and in some cases have experimental data for comparison. The close agreement of the **ANSYS** solutions to the theoretical or experimental results in this manual is intended to provide user confidence in the **ANSYS** solutions. These problems may then serve as the basis for additional validation and qualification of ANSYS capabilities by the user for specific applications that may be of interest.

The **ANSYS** software suite is continuously being verified by the developers (ANSYS, Inc.) as new capabilities are added to the programs. Verification of **ANSYS** products is conducted in accordance with written procedures that form a part of an overall Quality Assurance program at ANSYS, Inc. This manual represents a small subset of the Quality Assurance test case library which is used in full when testing new versions of **ANSYS FLUENT** and **ANSYS CFX**. This test library and the test cases in this manual represent comparisons of **ANSYS** solutions with known theoretical solutions, experimental results, or other independently calculated solutions. Since **ANSYS FLUENT** and **ANSYS CFX**are programs capable of solving very complicated practical engineering problems having no closed-form theoretical solutions, the relatively simple problems solved in this manual do not illustrate the full capability of these **ANSYS** programs.

In order to solve some test cases will require different product licenses; **ANSYS CFD**, **ANSYS FLUENT** or **ANSYS CFX**. If you do not have the appropriate licenses, you may not be able to reproduce the results.

1.1. Expected Results

The test cases in this manual have been modeled to give reasonably accurate comparisons with a low number of elements and iterations. In some cases, even fewer elements and/or iterations will still yield an acceptable accuracy. The test cases employ a balance between accuracy and solution time. An attempt has been made to present a test case and results that are grid independent. If test results are not grid independent, it is due to the need to limit the run time for the test to be in the manual. Improved results can be obtained in some cases by refining the mesh but requires longer solution times.

The ANSYS solutions in this manual are compared with solutions or experimental data from textbooks or technical publications. In some cases, the target (theoretical) answers reported in this manual may differ from those shown in the reference. In several fluid flow simulation problems where experimental results are available in the form of plots of the relevant parameters, the simulation results are also presented as plots so that the corresponding values can be compared on the same graph.

Many of the fluid dynamics simulation methods have to make use of data available from experimental measurements for their verification primarily because closed form theoretical solutions are not available for modeling the related phenomena. In this manual several test cases for **ANSYS FLUENT** and **ANSYS CFX** make use of experimental data published in reputed journals or conference proceedings for verification of the computational results. The experimental measurements for fluid flow systems are often presented in the form of plots of the relevant parameters. Hence the published experimental data for those cases and the corresponding simulation results are presented in graphical format to facilitate comparison.

Experimental data represent the “real world” physics reproduced in a controlled manner and provides more complex details of the flow field than theoretical solutions. The test cases in this manual have been modeled to give reasonably accurate comparisons with experimental data wherever applicable, with a low number of elements and iterations.

Different computers and operating systems may yield slightly different results for some of the test cases in this manual due to numerical precision variation from machine to machine. Solutions that are non-linear, iterative, or have convergence options activated are among the most likely to exhibit machine-dependent numerical differences. Because of this, an effort has been made to report an appropriate and consistent number of significant digits in both the target and the ANSYS solution. If you run these test cases on your own computer hardware, be advised that an ANSYS result reported in this manual as 0.01234 may very well show up in your printout as 0.012335271.

1.2. References

The goal for the test cases contained in this manual was to have results accuracy within 3% of the target solution. The solutions for the test cases have been verified, however, certain differences may exist with regard to the references. These differences have been examined and are considered acceptable.

It should be noted that only those items corresponding to the given theoretical solution values are reported for each problem. In most cases the same solution also contains a considerable amount of other useful numerical solution data.

Different computers and different operating systems may yield slightly different results for some of the test cases in this manual, since numerical precision varies from machine to machine. Because of this, an effort has been made to report an appropriate and consistent number of significant digits in both the target and the **ANSYS** solution. These results reported in this manual are from runs on an Intel Xeon processor using Microsoft Windows XP Professional. Slightly different results may be obtained when different processor types or operating systems are used.

1.3. Using the Verification Manual and Test Cases

You are encouraged to use these tests as starting points when exploring features in these products. Geometries, material properties, loads, and output results can easily be changed and the solution repeated. As a result, the tests offer a quick introduction to new features with which you may be unfamiliar.

The test cases in this manual are primarily intended for verification of the ANSYS programs. An attempt has been made to include most significant analysis capabilities of the ANSYS products in this manual. Although they are valuable as demonstration problems, the test cases are not presented as step-by-step examples with lengthy data input instructions and printouts. The reader should refer to the online help for complete input data instructions.

Users desiring more detailed instructions for solving problems or in-depth treatment of specific topics should refer to the suite of to the **ANSYS FLUENT** Documentation. **ANSYS FLUENT** Tutorials and **ANSYS CFX** Tutorials are also available for various specific topics. These publications focus on particular features or program areas, supplementing other **ANSYS** reference documents with theory, procedures, guidelines, examples, and references.

1.4. Quality Assurance Services

For customers who may have further need for formal verification of the ANSYS, Inc. products on their computers, ANSYS, Inc. offers the Quality Assurance Testing Agreement. The user is provided with input data, output data, comparator software, and software tools for automating the testing and reporting process. If you are interested in contracting for such services, contact the ANSYS, Inc. Quality Assurance Group.

1.5. Index of ANSYS Fluid Dynamics Test cases

Dimensionality Column Key:

- **2** -- 2-D
- **3** -- 3-D
- **A** -- 2-D Axisymmetric

	Dimensionality	Unsteady	Turbulent	Trans. Turbulence	Inviscid	Compressible	Non-Isothermal	Radiation	Multi-Species	Reacting	Multi-phase	Phase-Change	External Forces	Moving Frame
VMFL001	2													
VMFL002	A						X							
VMFL003	A		X											
VMFL004	2													
VMFL005	A													
VMFL006	A								X					
VMFL007	A													
VMFL008	A													X
VMFL009	2						X						X	
VMFL010	2													
VMFL011	2													
VMFL012	2		X											
VMFL013	2		X				X							
VMFL014	A		X						X					
VMFL015	3		X											
VMFL016	3		X											
VMFL017	2		X			X	X							
VMFL018	2		X			X	X							
VMFL019	2	X												
VMFL020	2	X					X							X
VMFL021	A		X								X	X		
VMFL022	A		X								X	X		

	Dimensionality	Unsteady	Turbulent	Trans. Turbulence	Inviscid	Compressible	Non-Isothermal	Radiation	Multi-Species	Reacting	Multi-phase	Phase-Change	External Forces	Moving Frame
VMFL023	2	X												
VMFL024	A	X									X		X	
VMFL025	A		X				X		X	X				
VMFL026	3	X			X	X	X							
VMFL027	2		X											
VMFL028	A		X				X							
VMFL029	2				X		X							
VMFL030	3		X											
VMFL031	2		X											
VMFL032	A		X											
VMFL033	2						X							
VMFL034	2		X								X			
VMFL035	3		X			X	X							X
VMFL036	A		X											
VMFL037	2		X											
VMFL038	2	X											X	
VMFL039	A		X				X				X	X	X	
VMFL040	A		X											
VMFL041	2		X			X	X							
VMFL042	2		X								X		X	
VMFL043	2		X	X										
VMFL044	A		X			X	X							
VMFL045	2					X	X							
VMFL046	2				X	X	X							
VMFL047	2		X											
VMFL048	3		X											
VMFL049	A		X				X		X	X				
VMFL050	2	X					X							
VMFL051	2				X	X	X							
VMFL052	2		X										X	
VMFL053	2		X			X	X							
VMFL054	2													
VMFL055	2		X	X									X	
VMFL056	2						X	X						

	Dimensionality	Unsteady	Turbulent	Trans. Turbulence	Inviscid	Compressible	Non-Isothermal	Radiation	Multi-Species	Reacting	Multi-phase	Phase-Change	External Forces	Moving Frame
VMFL057	2						X	X						
VMFL058	A		X											
VMFL059	2						X							
VMFL060	2		X	X		X	X							
VMFL061	2						X	X						
VMFL062	2		X											
VMFL063	2													
VMFL064	2													
VMFL065	A		X											
VMFL066	2						X	X						

VMFL001: Flow Between Rotating and Stationary Concentric Cylinders

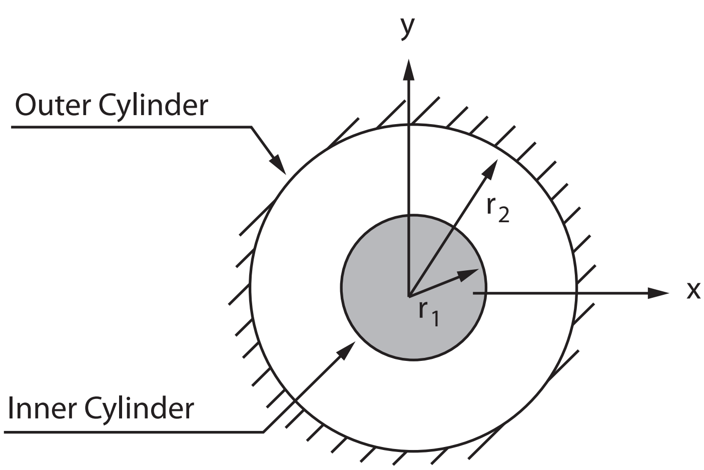
Overview

Reference	F. M. White. "*Viscous Fluid Flow*". Section 3-2.3. McGraw-Hill Book Co., Inc.. New York, NY. 1991.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Laminar flow, rotating wall
Input Files	**`rot_conc_cyl.cas`** for ANSYS FLUENT **`rotating_cylinder.def`** for ANSYS CFX

Test Case

Steady laminar flow between two concentric cylinders is modeled. The flow is induced by rotation of the inner cylinder with a constant angular velocity, while the outer cylinder is held stationary. Due to periodicity only a section of the domain needs to be modeled. In the present simulation a 180° segment (half of the domain shown in *Figure 1* (p. 9)) is modeled. The sketch is not to scale.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density = 1 kg/m^3 Viscosity = 0.002 kg/m-s	Radius of the Inner Cylinder = 17.8 mm Radius of the outer Cylinder = 46.8 mm	Angular velocity of the inner wall = 1 rad/s

Analysis Assumptions and Modeling Notes

The flow is steady. The tangential velocity at various sections can be calculated using analytical equations for laminar flow. These values are used for comparison with simulation results.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Tangential Velocity in the Annulus at Various Radial Locations

Tangential Velocity at	Target Calculation, m/s	ANSYS FLUENT, m/s	Ratio
r = 20 mm	0.0151	0.0151	0.999
r = 25 mm	0.0105	0.0105	0.996
r = 30 mm	0.0072	0.0072	0.990
r = 35 mm	0.0046	0.0045	0.979

Results Comparison for ANSYS CFX

Table 2 Comparison of Tangential Velocity in the Annulus at Various Radial Locations

Location	Target Calculation, m/s	ANSYS CFX, m/s	Ratio
r = 20 mm	0.0151	0.0150	0.991
r = 25 mm	0.0105	0.0105	0.998
r = 30 mm	0.0072	0.0071	0.988
r = 35 mm	0.0046	0.0045	0.976

VMFL002: Laminar Flow Through a Pipe with Uniform Heat Flux

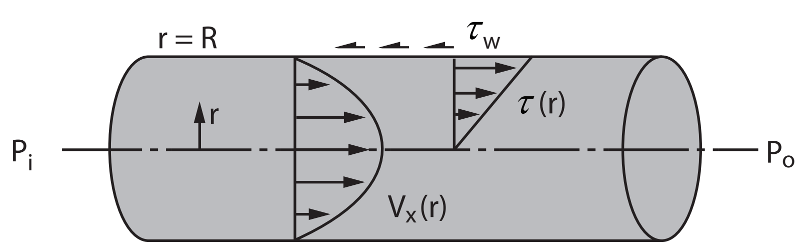
Overview

Reference	F. M. White. "*Fluid Mechanics* ". 3rd Edition. McGraw-Hill Book Co.. New York, NY. 1994. F. P. Incropera and D. P. DeWitt. "*Fundamentals of Heat Transfer*". John Wiley & Sons. 1981.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Laminar flow with heat transfer
Input File	**`laminar-pipe-hotflow.cas`** for ANSYS FLUENT **`VMFL002B_VV002CFX.def`** for ANSYS CFX

Test Case

Laminar flow of Mercury through a circular pipe is modeled, with uniform heat flux across the wall. A fully developed laminar velocity profile is prescribed at the inlet. The resulting pressure drop and exit temperature are compared with analytical calculations for Laminar flow. Only half of the 2–D domain is modeled due to symmetry.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Fluid: Mercury Density = 13529 kg/m^3 Viscosity = 0.001523 kg/m-s Specific Heat = 139.3 J/kg-K Thermal Conductivity = 8.54 W/m-K	Length of the pipe = 0.1 m Radius of the pipe = 0.0025 m	Fully developed velocity profile at inlet. Inlet temperature = 300 K Heat Flux across wall = 5000 W/m^2

Analysis Assumptions and Modeling Notes

The flow is steady and incompressible. Pressure drop can be calculated from the theoretical expression for laminar flow given in Ref. 1. Correlations for temperature calculations are given in Ref. 2.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Pressure Drop and Outlet Temperature

	Target Calculation	ANSYS FLUENT	Ratio
Pressure Drop	1.000 Pa	0.999 Pa	0.999
Centerline Temperature at the Outlet	341.00 K	340.50 K	0.999

Results Comparison for ANSYS CFX

Table 2 Comparison of Pressure Drop and Outlet Temperature

	Target Calculation	ANSYS CFX	Ratio
Pressure Drop	1.000 Pa	1.019 Pa	1.019
Centerline Temperature at the Outlet	341.00 K	340.8 K	0.9994

VMFL003: Pressure Drop in Turbulent Flow through a Pipe

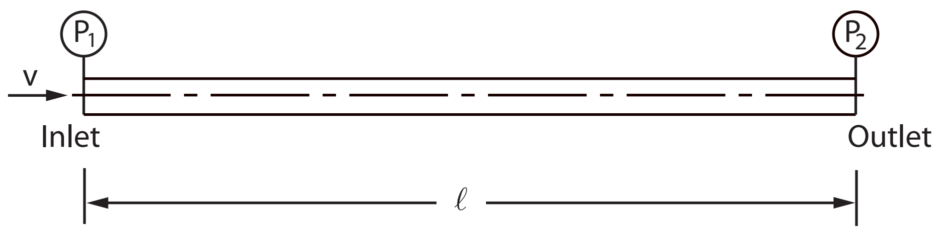
Overview

Reference	F. M. White. "*Fluid Mechanics*". 3rd Edition. McGraw-Hill Co.. New York, NY. 1994.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent flow, standard k-ε Model
Input File	**`turb_pipe_flow.cas`** for ANSYS FLUENT **`VMFL003B_VV003CFX.def`** for ANSYS CFX

Test Case

Air flows through a horizontal pipe with smooth walls. The flow Reynolds number is 1.37 X 10^4. Only half of the axisymmetrical domain is modeled.

Figure 1 Flow Domain



The figure is not to scale.

Material Properties	Geometry	Boundary Conditions
Density = 1.225 kg/m^3 Viscosity = 0.001523 kg/m-s	Length of the pipe = 2 m Radius of the pipe = 0.002 m	Inlet velocity = 50 m/s Outlet pressure = 0 Pa

Analysis Assumptions and Modeling Notes

The flow is steady. Pressure drop can be calculated from analytical formula using friction factor f which can be determined for the given Reynolds number from Moody chart. The calculated pressure drop is compared with the simulation results (pressure difference between inlet and outlet).

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Pressure Drop in the Pipe

	Target Calculation	ANSYS FLUENT	Ratio
Pressure Drop	21789 Pa	21480 Pa	0.988

Results Comparison for ANSYS CFX

Table 2 Comparison of Pressure Drop in the Pipe

	Target Calculation	ANSYS CFX	Ratio
Pressure Drop	21789 Pa	21740 Pa	0.9975

VMFL004: Plain Couette Flow with Pressure Gradient

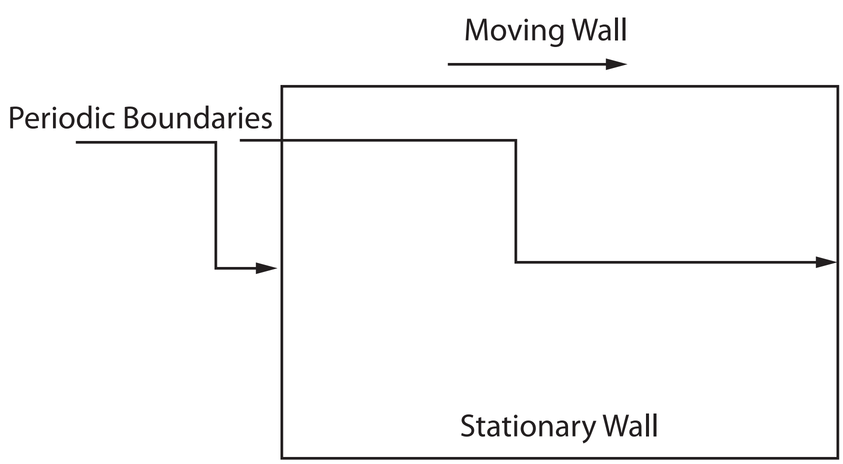
Overview

Reference	Munon, Young, Okiishi. "*Fundamentals of Fluid Mechanics*". 5th Edition.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Laminar flow, moving wall, periodic boundaries
Input Files	`couette_flow.cas` for ANSYS FLUENT `Couette_Flow.def` for ANSYS CFX

Test Case

Viscous flow between two parallel plates is modeled. The top plate moves with a uniform velocity while the lower plate is fixed. A pressure gradient is imposed in a direction parallel to the plates.

Figure 1 Flow Domain



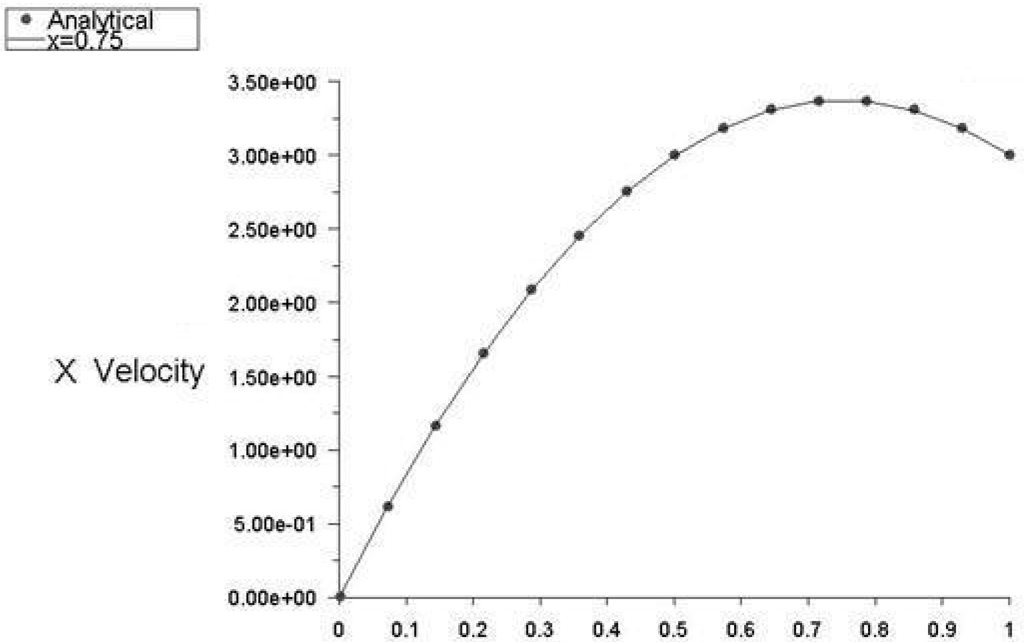
Material Properties	Geometry	Boundary Conditions
Density = 1 kg/m^3 Viscosity = 1 kg/m-s	Length of the domain = 1.5 m Width of the domain = 1 m	Velocity of the moving wall = 3 m/s in X-direction For ANSYS FLUENT, pressure gradient across periodic boundaries = -12 Pa/m For ANSYS CFX, pressure gradient across periodic boundaries = -12 Pa/m (pressure change = –18 Pa)

Analysis Assumptions and Modeling Notes

The flow is steady and laminar. Periodic conditions with specified pressure drop are applied across the flux boundaries.

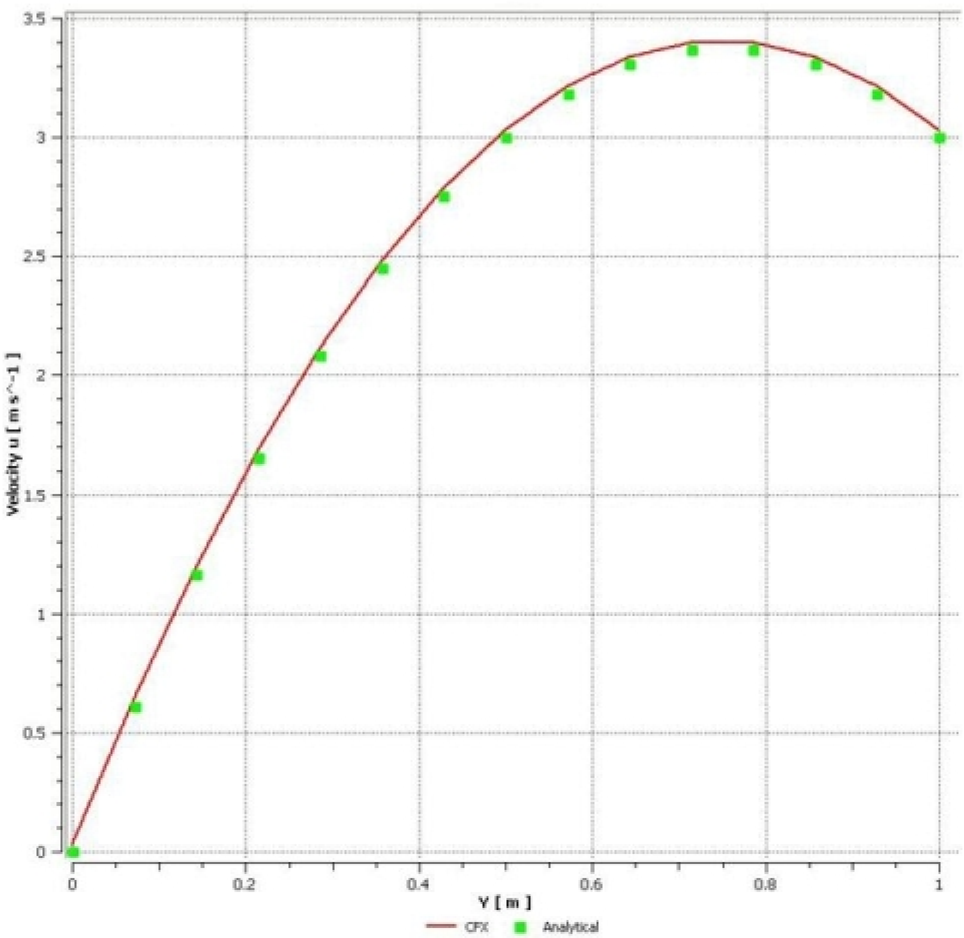
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of X-Velocity (m/s) at a Section Where X = 0.75 m



Results Comparison for ANSYS CFX

Figure 3 Comparison of X-Velocity (m/s) at a Section Where X = 0.75 m



VMFL005: Poiseuille Flow in a Pipe

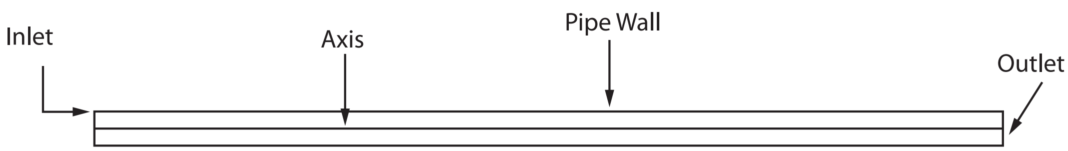
Overview

Reference	F. M. White. "*Fluid Mechanics*". 3rd Edition. McGraw-Hill Book Co.. New York, NY. 1994.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Steady laminar flow
Input File	`poiseuille-flow.cas` for ANSYS FLUENT `VMFL005B_VV005CFX.def` for ANSYS CFX

Test Case

Fully developed laminar flow in a circular tube is modeled. Reynolds number based on the tube diameter is 500. Only half of the axisymmetric domain is modeled.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density = 1 kg/m^3 Viscosity = 1e-5 kg/m-s	Length of the pipe = 0.1 m Radius of the pipe = 0.00125 m	Fully developed laminar velocity profile at inlet with an average velocity of 2.00 m/s

Analysis Assumptions and Modeling Notes

The flow is steady. A fully developed laminar velocity profile is prescribed at the inlet. Hagen-Poiseuille equation is used to determine the pressure drop analytically.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Pressure Drop in the Pipe

	Target Calculation	ANSYS FLUENT	Ratio
Pressure Drop	10. 24 Pa	10.22 Pa	0.998

Results Comparison for ANSYS CFX

Table 2 Comparison of Pressure Drop in the Pipe

	Target Calculation	ANSYS CFX	Ratio
Pressure Drop	10. 24 Pa	10.49 Pa	1.024

VMFL006: Multicomponent Species Transport in Pipe Flow

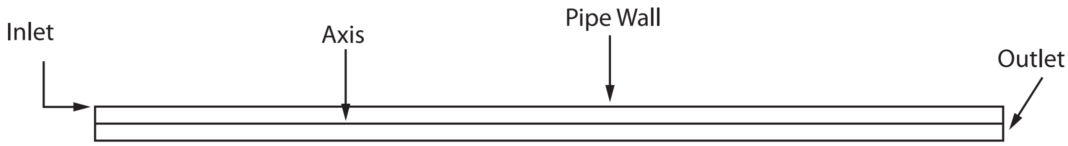
Overview

Reference	W. M. Kays and M. E. Crawford. "*Convective Heat and Mass Transfer*". 3rd Edition. McGraw-Hill Book Co., Inc.. New York, NY. 126-134. 1993.
Solver	ANSYS FLUENT (ANSYS CFX simulation is not available for this case)
Physics/Models	Steady laminar flow, species transport
Input File	`Species-diffusion.cas`

Test Case

Fully developed laminar flow in a circular tube, with two species is modeled. Species A enters at the inlet and species B enters from the wall. Uniform and dissimilar mass fractions are specified at the pipe inlet and wall. Fluid properties are assumed to be the same for both species, so that computed results can be compared with analytical solution.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Species A Density = 1 kg/m^3 Viscosity = 1.0×10^{-5} Pa-s Diffusivity $_{BA}$ = 1.43×10^{-5} m^2/s **Species B** Density = 1 kg/m^3 Viscosity = 1.0×10^{-5} Pa-s Diffusivity $_{AB}$ = 1.43×10^{-5} m^2/s	Length of the pipe = 0.1 m Radius of the pipe = 0.0025 m	Fully developed laminar velocity profile at inlet with an average velocity of 1 m/s Mass fraction of species A at pipe inlet = 1.0 Mass fraction of species B at pipe inlet = 0.0 Mass fraction of species A at pipe wall = 0.0 Mass fraction of species B at pipe wall = 1.0

Analysis Assumptions and Modeling Notes

The flow is steady. A fully developed laminar velocity profile is prescribed at the inlet. Species transport model is used.

Results Comparison

Table 1 Comparison of Mass Fraction of Species A Along the Axis

Axial location (m)	Target Calculation	ANSYS FLUENT	Ratio
0.01	0.8225	0.8223	1.000
0.02	0.7308	0.7307	1.000
0.03	0.6593	0.6592	1.000
0.04	0.5992	0.5991	1.000
0.05	0.5469	0.5469	1.000
0.06	0.5006	0.5006	1.000
0.07	0.4589	0.4591	1.000
0.08	0.4212	0.4214	1.000
0.09	0.3869	0.3871	1.001
0.10	0.3555	0.3558	1.001

VMFL007: Non-Newtonian Flow in a Pipe

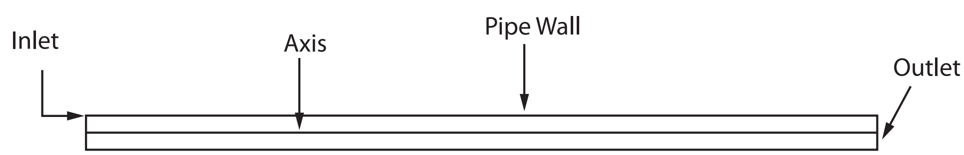
Overview

Reference	W. F. Hughes and J. A. Brighton. "*Schaum's Outline of Theory and Problems of Fluid Dynamics*." McGraw-Hill Book Co., Inc.. New York, NY. 1991.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Steady laminar flow, power law for viscosity
Input File	**`powerlaw-visc.cas`** for ANSYS FLUENT **`VMFL007B_vv007CFX.def`** for ANSYS CFX

Test Case

Flow of a non-Newtonian fluid in a circular pipe is modeled. Viscosity is specified by power law equation.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density = 1000 kg/m^3 Viscosity: Power law Parameters: k = 10 n = 0.4	Pipe length = 0.1 m Pipe diameter = 0.0025 m	Fully developed velocity profile at inlet with an average velocity of 2 m/s

Analysis Assumptions and Modeling Notes

The flow is steady. Viscosity is specified using non-Newtonian power law equation.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Pressure Drop in the Pipe

	Target Calculation	**ANSYS FLUENT**	**Ratio**
Pressure Drop	60.52 kPa	60.37 kPa	0.998

Results Comparison for ANSYS CFX

Table 2 Comparison of Pressure Drop in the Pipe

	Target Calculation	ANSYS CFX	Ratio
Pressure Drop	60.52 kPa	61.52 kPa	1.0165

VMFL008: Flow Inside a Rotating Cavity

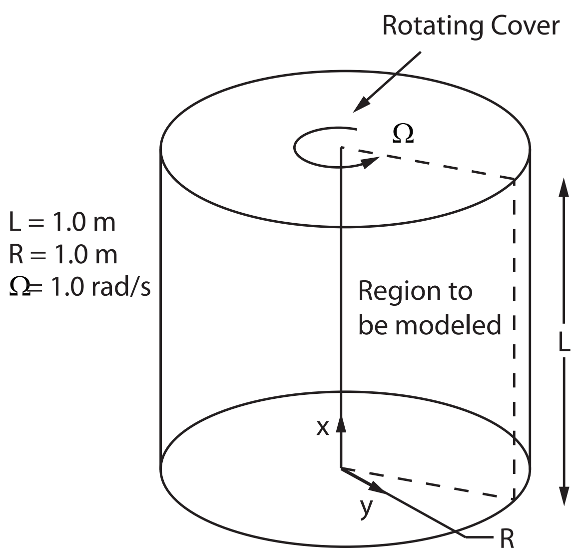
Overview

Reference	J. A. Michelsen. "*Modeling of Laminar Incompressible Rotating Fluid Flow*". *AFM 86-05*., Ph.D. thesis. Department of Fluid Mechanics, Technical University of Denmark. 1986.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Laminar flow, Rotating reference frame
Input File	`rotcv_RRF.cas` for ANSYS FLUENT `VMFL008B_rot_cyl.def` for ANSYS CFX

Test Case

Flow in a cylindrical cavity enclosed with a lid that spins at Ω = **1.0** rad/s. The flow field is 2–D axisymmetric, so only the region bounded by the dashed lines in *Figure 1* (p. 25)needs to be modeled. The Reynolds number of the flow based on the cavity radius **R** and the tip-speed of the disk is **1800**.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density = 1 kg/m^3 Viscosity: 0.000556 kg/m-s	Height of the cavity = 1m Radius of cavity = 1m	Speed of rotation of the moving wall = 1rad/s Rotational velocity for cell zone = -1rad/s

Analysis Assumptions and Modeling Notes

The flow is laminar. The problem is solved using rotating reference frame.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Distribution of Radial Velocity Along a Section at X= 0.6 m

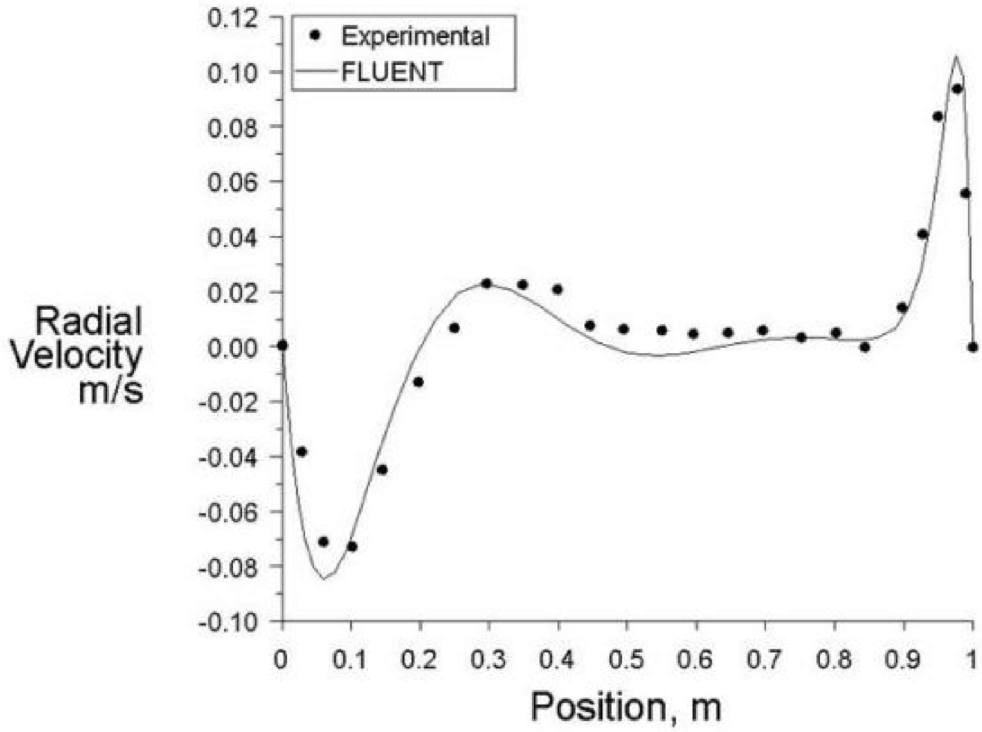
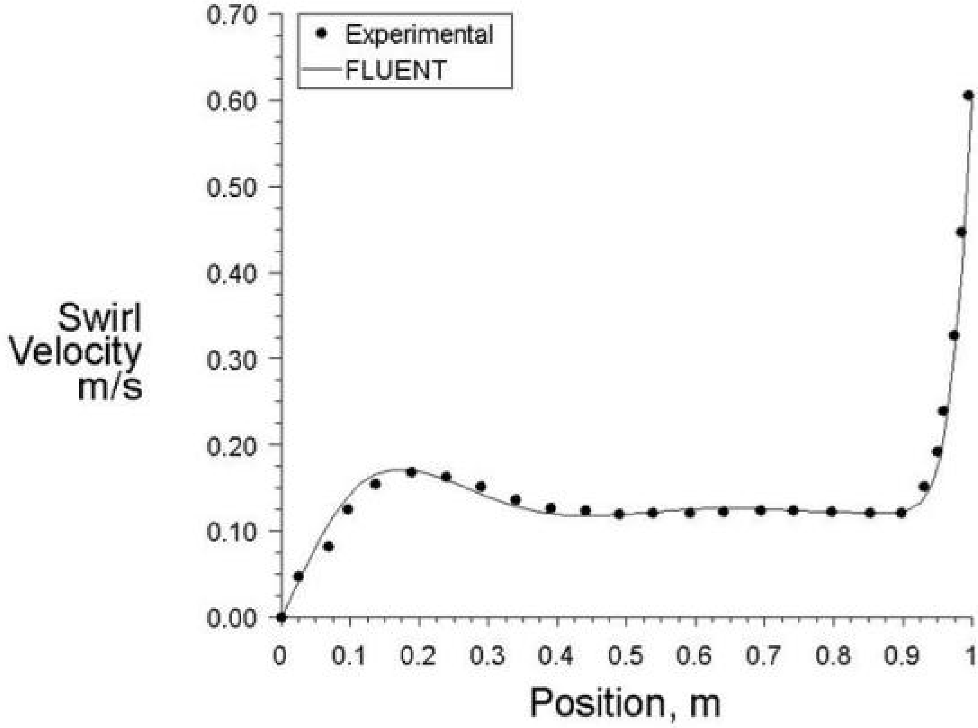


Figure 3 Comparison of Distribution of Swirl Velocity Along a Section at X= 0.6 m



Results Comparison for ANSYS CFX

Figure 4 Comparison of Distribution of Radial Velocity Along a Section at X= 0.6 m

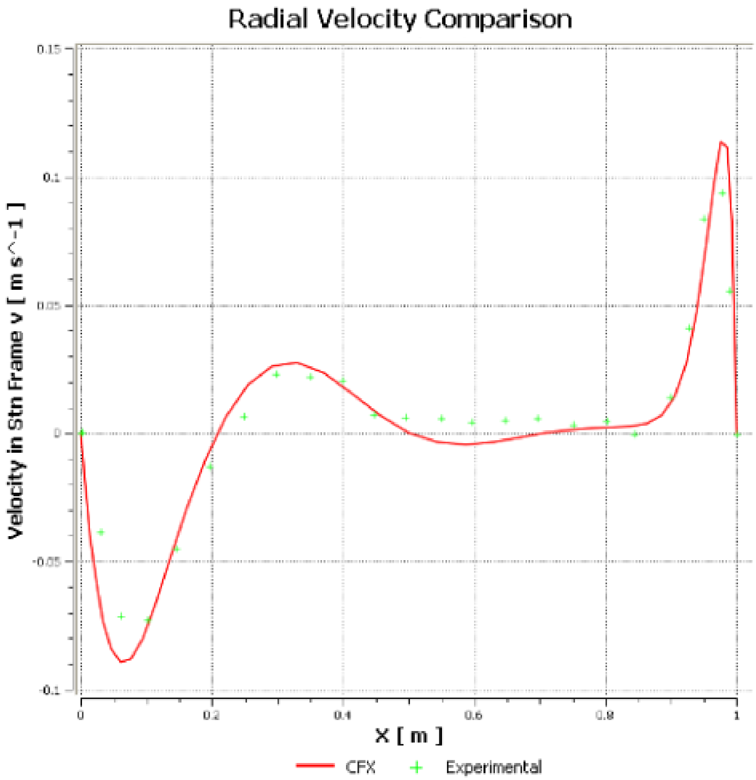
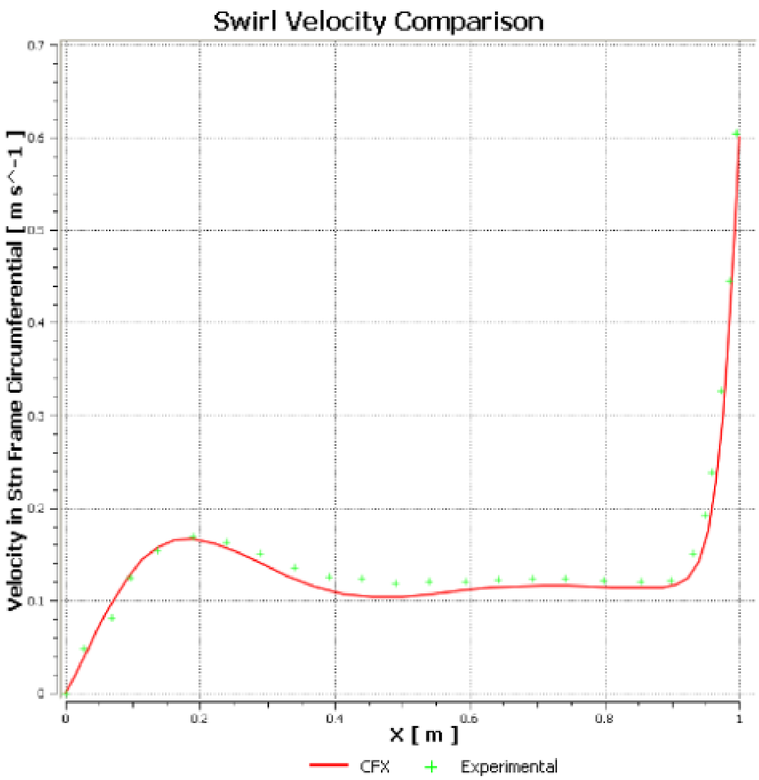


Figure 5 Comparison of Distribution of Swirl Velocity Along a Section at X= 0.6 m



VMFL009: Natural Convection in a Concentric Annulus

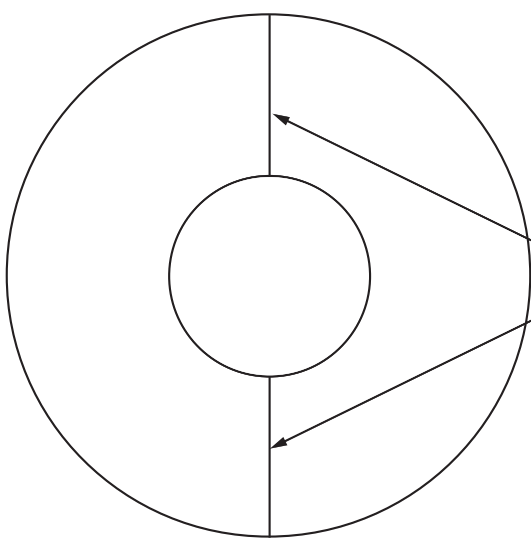
Overview

Reference	Kuehn, T.H. and Goldstein, R.J., An Experimental Study of Natural Convection Heat Transfer in Concentric and Eccentric Horizontal Cylindrical Annuli, *Journal of Heat Transfer*, 100:635-640, 1978.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Heat transfer, natural convection, laminar flow
Input Files	`concn.cas` for ANSYS FLUENT `ecc_cfx.def` for ANSYS CFX

Test Case

Natural convection inside a concentric annular domain. The inner wall is maintained at a higher temperature than the outer wall, thereby causing buoyancy induced circulation.

Figure 1 Flow Domain



Only half of the domain is modeled due to symmetry.

Material Properties	Geometry	Boundary Conditions
Density: Incompressible ideal gas Viscosity: 2.081 X 10^{-5} kg/m-s Specific Heat: 1008 J/kg-K	Radius of outer cylinder = 46.25 mm Radius of inner cylinder = 17.8 mm	Inner wall temperature = 373 K Outer wall temperature = 327 K

Material Properties	Geometry	Boundary Conditions
Thermal Conductivity: 0.02967 W/m-K		

Analysis Assumptions and Modeling Notes

The flow is symmetric and only half of the domain is modeled. Density is calculated based on incompressible ideal gas assumption. The flow is laminar.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Static Temperature Distribution on the Bottom Wall of Symmetry

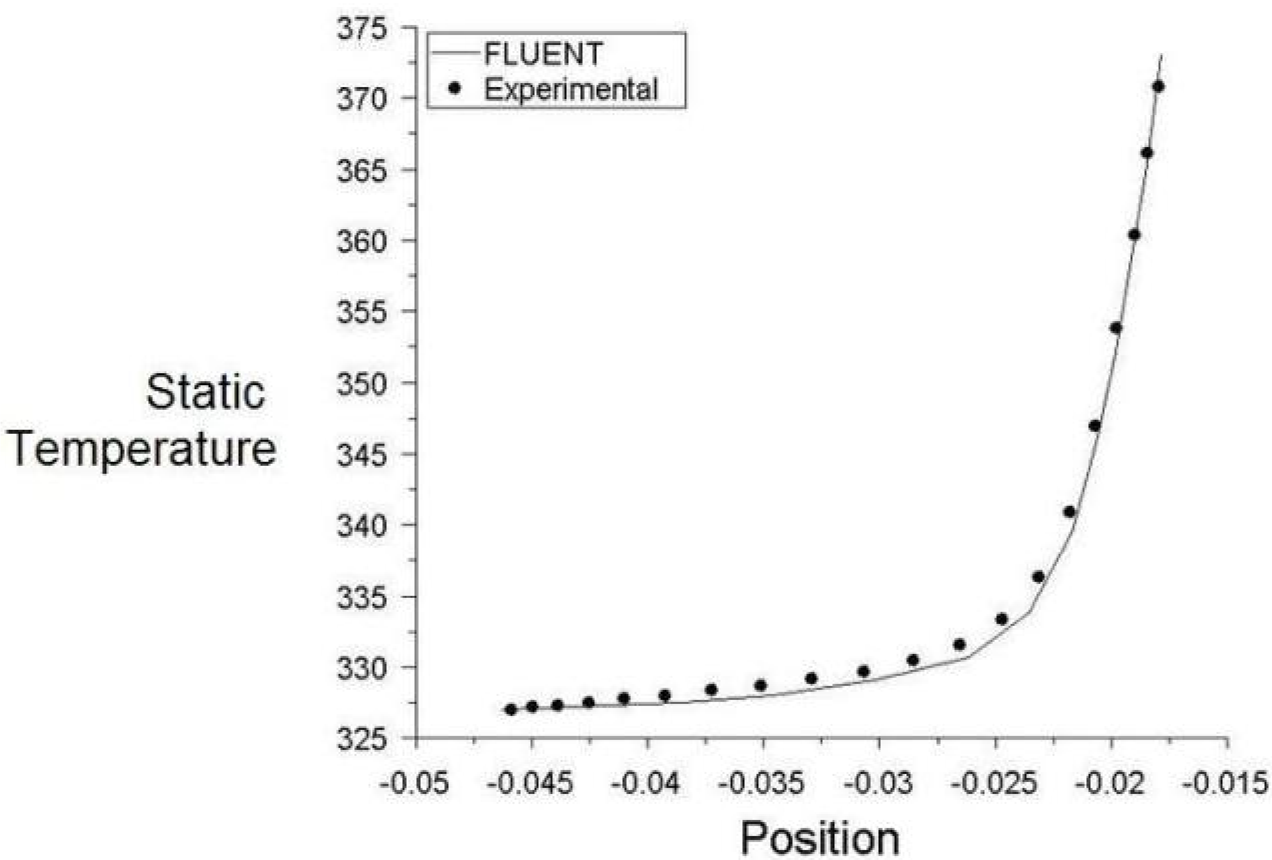
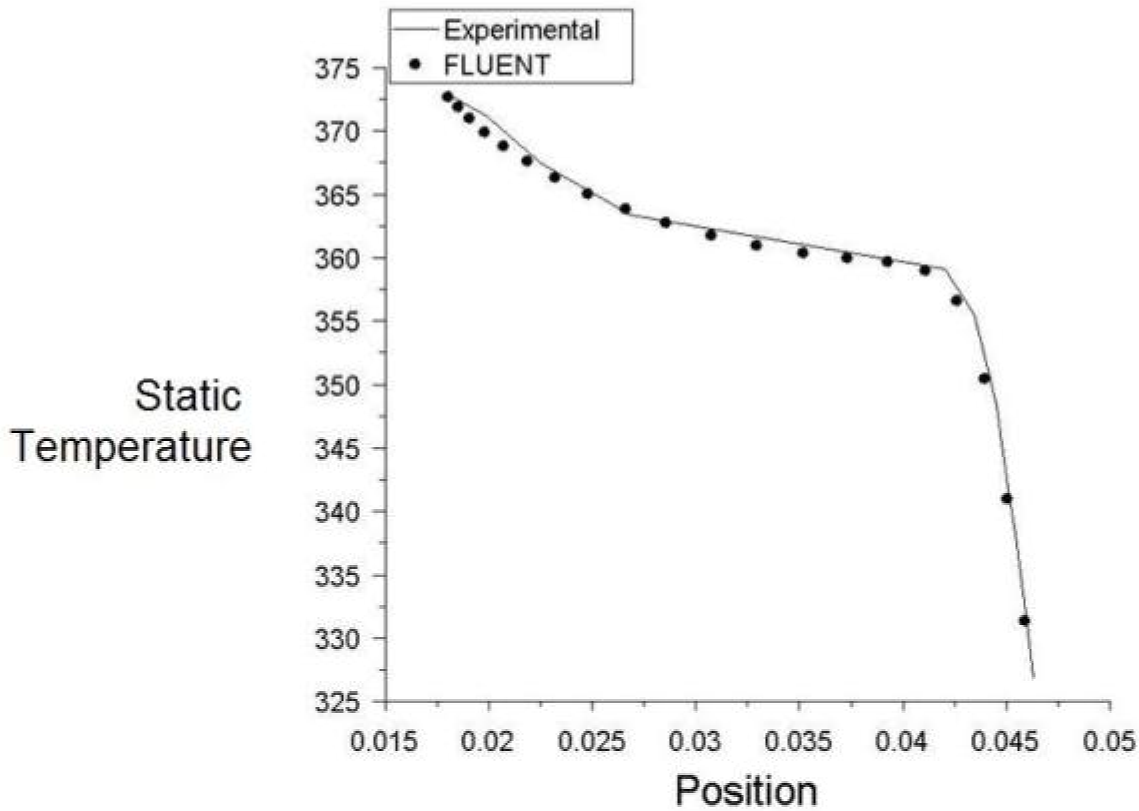
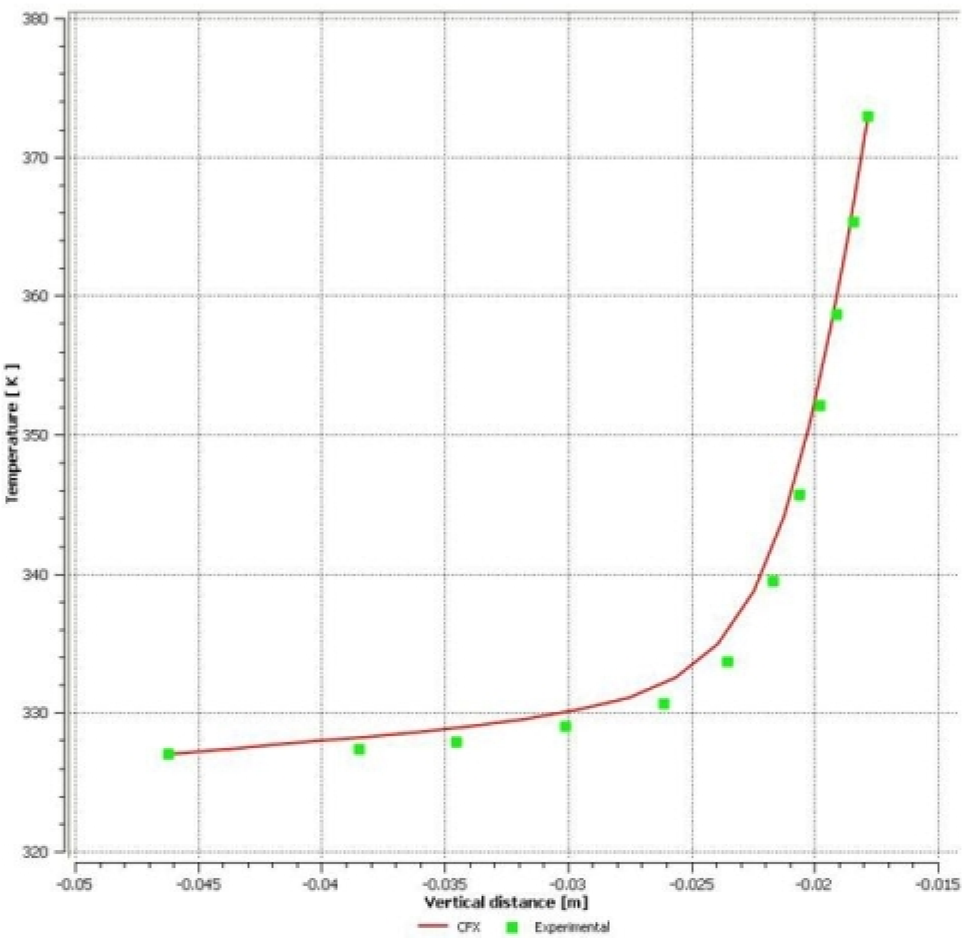


Figure 3 Comparison of Static Temperature Distribution on the Top Wall of Symmetry



Results Comparison for ANSYS CFX

Figure 4 Comparison of Static Temperature Distribution on the Bottom Wall of Symmetry



VMFL010: Laminar Flow in a 90° Tee-Junction.

Overview

Reference	R.E. Hayes, K. Nandkumar, and H. Nasr-El-Din. "*Steady Laminar Flow in a 90 Degree Planar Branch*". *Computers and Fluids*, 17(4). 537-553. 1989.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Laminar flow
Input File	**`plarb_r4.cas`** for ANSYS FLUENT **`VMFL010B_plarb.def`** for ANSYS CFX

Test Case

The purpose of this test is to compare prediction of the fractional flow in a dividing tee-junction with experimental results. The fluid enters through the bottom branch and divides into the two channels whose exit planes are held at the same static pressure.

Figure 1 Flow Domain

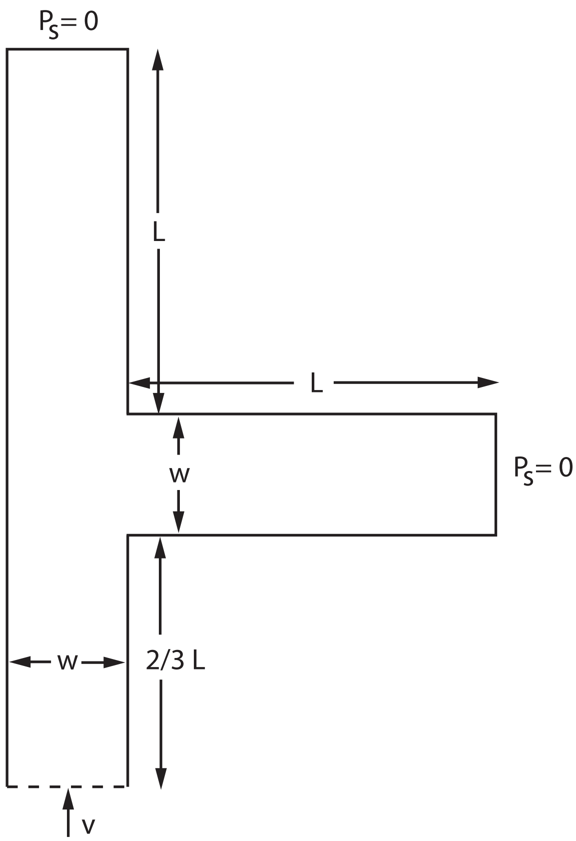


Table 1 Comparison of Flow Split from Tee

Material Properties	Geometry	Boundary Conditions
Fluid: Air Density : 1 kg/m^3 Viscosity: 0.003333 kg/m-s	L=3.0 m W=1.0 m	Fully developed inlet velocity profile for: $\mathrm{Re}=\frac{\rho V_c W}{\mu}=300$ where V_c is the inlet centerline velocity. $P_s=0$

Analysis Assumptions and Modeling Notes

The flow is steady and incompressible. Pressure based solver is used. It is seen that with increasing flow rate in the main channel, less fluid escapes through the secondary (right) branch. For analysis of results, we calculate and compare the fractional flow in the upper branch.

Results Comparison for ANSYS FLUENT

Table 2 Comparison of Flow Split from Tee

	Target	ANSYS FLUENT	Ratio
Flow split	0.887	0.884	0.997

Results Comparison for ANSYS CFX

Table 3 Comparison of Flow Split from Tee

	Target	ANSYS CFX	Ratio
Flow split	0.887	0.8837	0.9962

VMFL011: Laminar flow in a Triangular Cavity

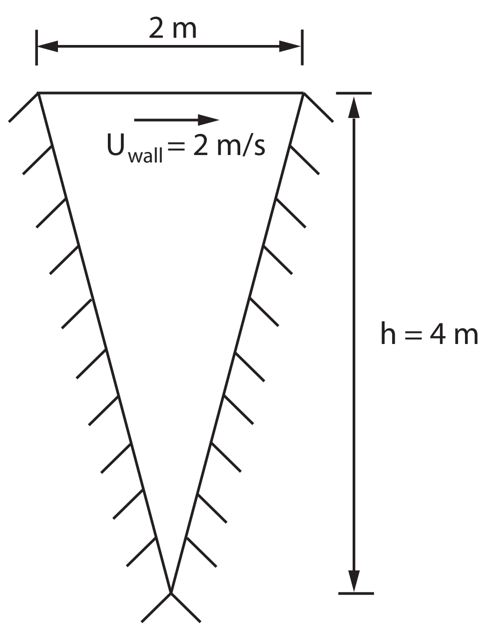
Overview

Reference	R. Jyotsna and S.P. Vanka. "*Multigrid Calculation of Steady, Viscous Flow in a Triangular Cavity*". *J. Comp. Phys.*, 122. 107-117. 1995.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Viscous flow, driven by a moving wall
Input Files	`driv.cas` for FLUENT `driven_cavity.def` for ANSYS CFX

Test Case

Laminar flow induced by the motion of the top wall of a triangular cavity (*Figure 1* (p. 39)). The side walls are stationary.

Figure 1 Flow Domain



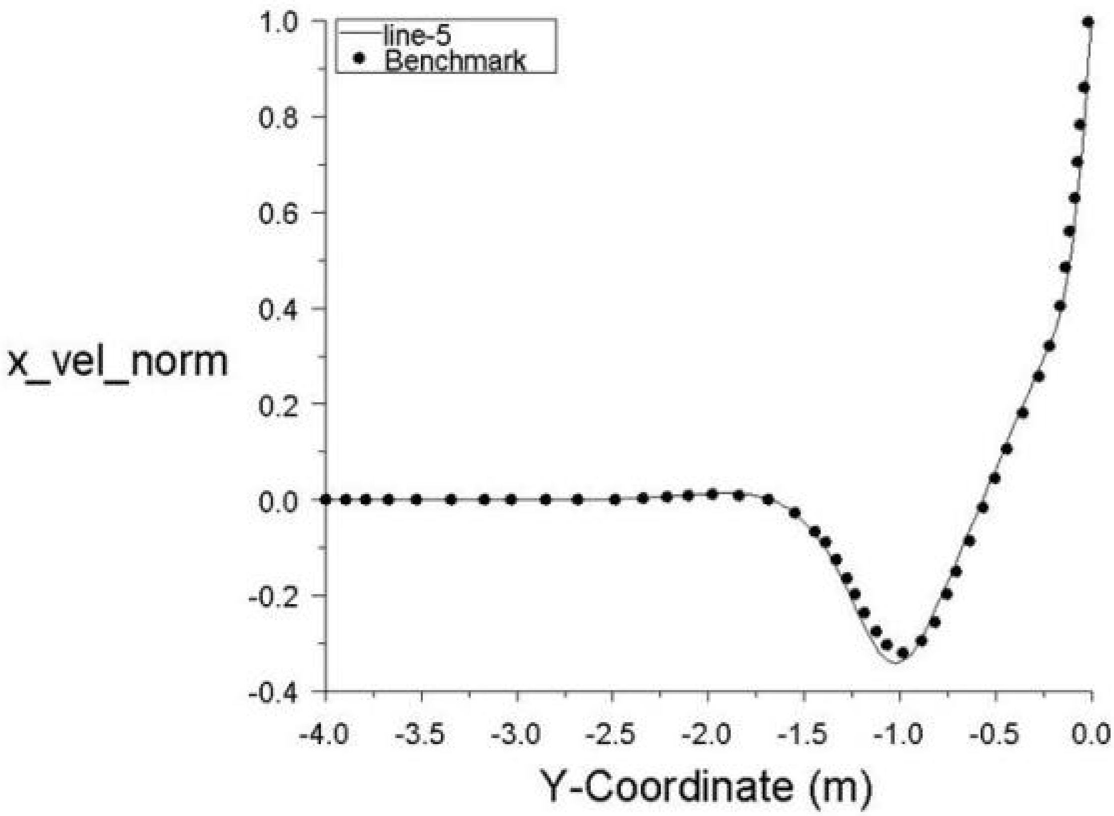
Material Properties	Geometry	Boundary Conditions
Density = 1 kg/m^3 Viscosity = 0.01 kg/m-s	Height of the triangular cavity = 4 m Width of the base = 2 m	Velocity of the top (base) wall = 2 m/s Other walls are stationary

Analysis Assumptions and Modeling Notes

The flow is steady. Pressure based solver is used. A hybrid mesh with triangular and quadrilateral cells is used to discretize the domain.

Results Comparison for FLUENT

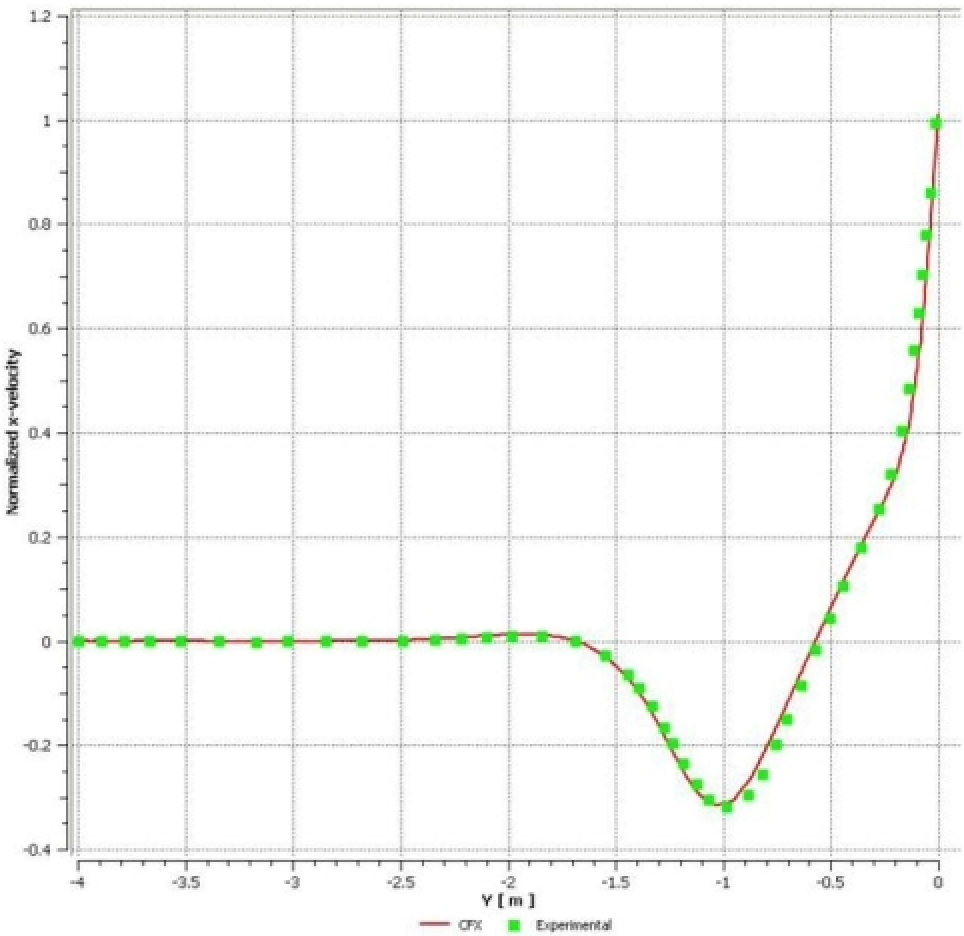
Figure 2 Comparison of Distribution of Normalized X-Velocity Along a Vertical Line that Bisects the Base of the Cavity



In this figure, X-velocity is normalized by the velocity of the moving wall.

Results Comparison for ANSYS CFX

Figure 3 Comparison of Distribution of Normalized X-Velocity Along a Vertical Line that Bisects the Base of the Cavity



In this figure also the X-velocity is normalized by the velocity of the moving wall.

VMFL012: Turbulent Flow in a Wavy Channel

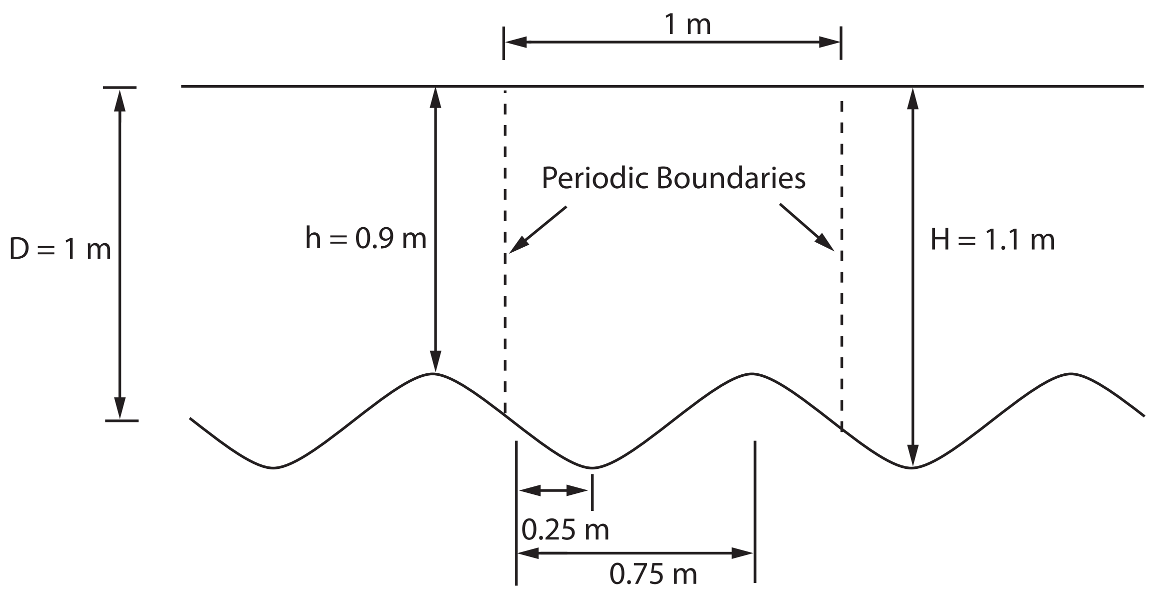
Overview

Reference	J.D. Kuzan. "*Velocity Measurements for Turbulent Separated and Near-Separated Flows Over Solid Waves*". Ph.D. thesis. Dept. Chem. Eng., Univ. Illinois. Urbana, IL. 1986.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent internal flow with separation and recirculation, periodic boundaries
Input File	`wavy.cas` for ANSYS FLUENT `VMFL012B_VV012.def` for ANSYS CFX

Test Case

A periodic flow domain bounded on one side by a sinusoidal wavy wall and with a straight wall on the other side. Due to periodicity only a part of the channel needs to modeled. *Figure 1* (p. 43) depicts the channel geometry. Flow direction is from left to right.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density = 1 kg/m^3 Viscosity = 0.0001 kg/m-s	Amplitude of the sinusoidal wave = 0. 1m Wave length = 1 m Length of the periodic segment = 1 m	Periodic Conditions: Mass flow rate = 0.816 kg/S Pressure Gradient = -0.01687141 Pa/m

Analysis Assumptions and Modeling Notes

The flow is steady. Pressure based solver is used. Periodic boundaries are used. For analysis of results, velocity in the x –direction is normalized by the mean mainstream velocity, U = 0.816 m/s, at mean channel height.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Distribution of Normalized X-Velocity along Transverse Direction at the Wave Crest

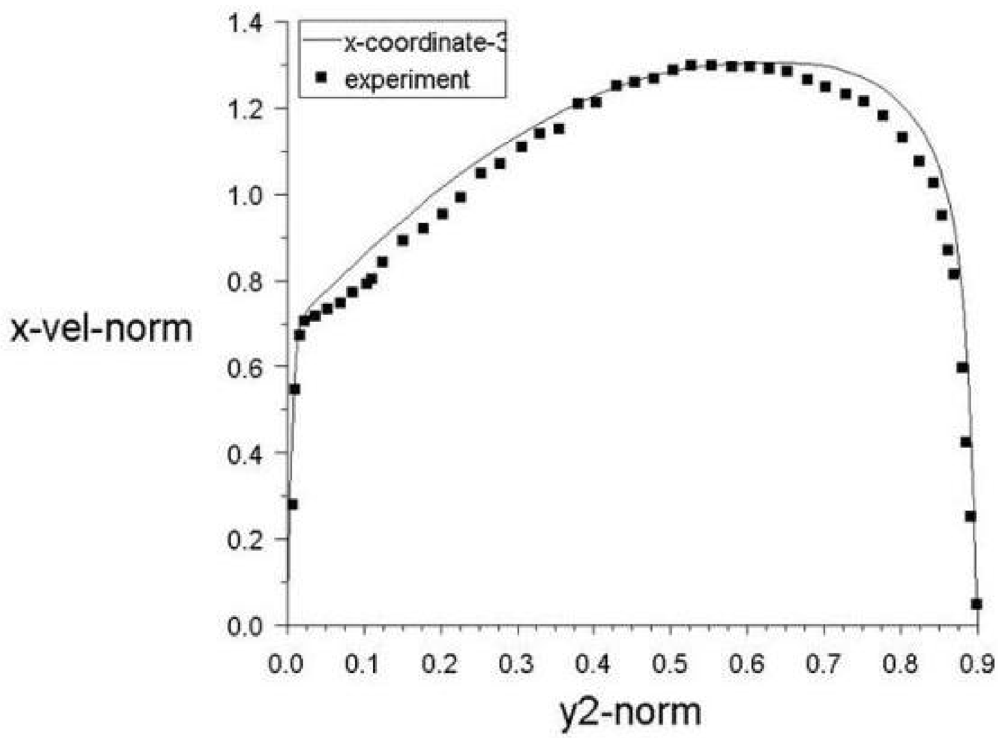
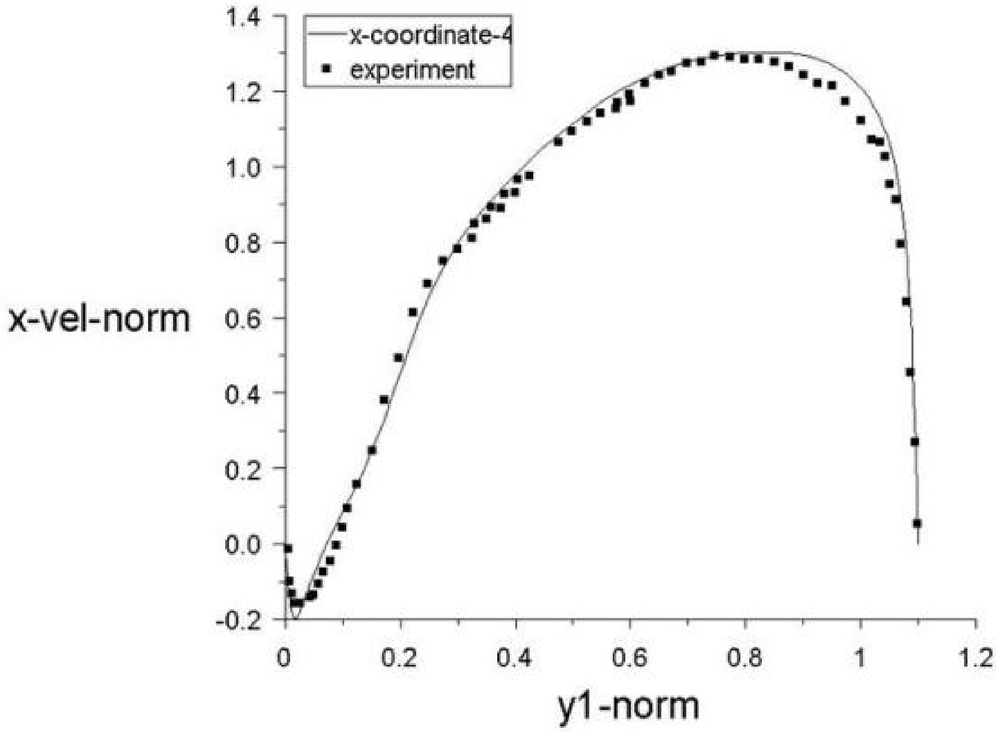


Figure 3 Comparison of Predicted Normalized X-Velocity along Transverse Direction at the Wave Trough



Results Comparison for ANSYS CFX

Figure 4 Comparison of Distribution of Normalized X-Velocity along Transverse Direction at the Wave Crest

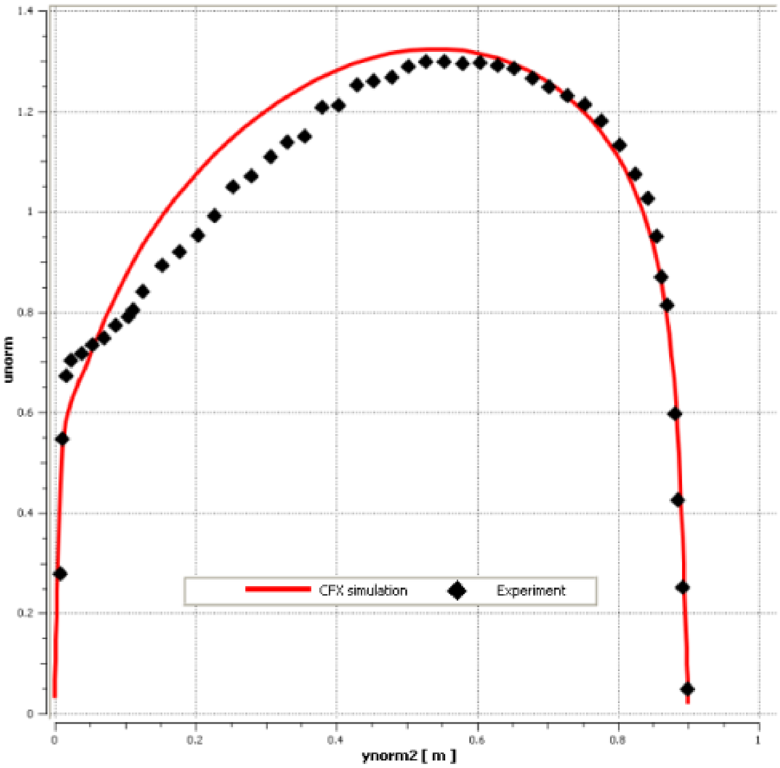
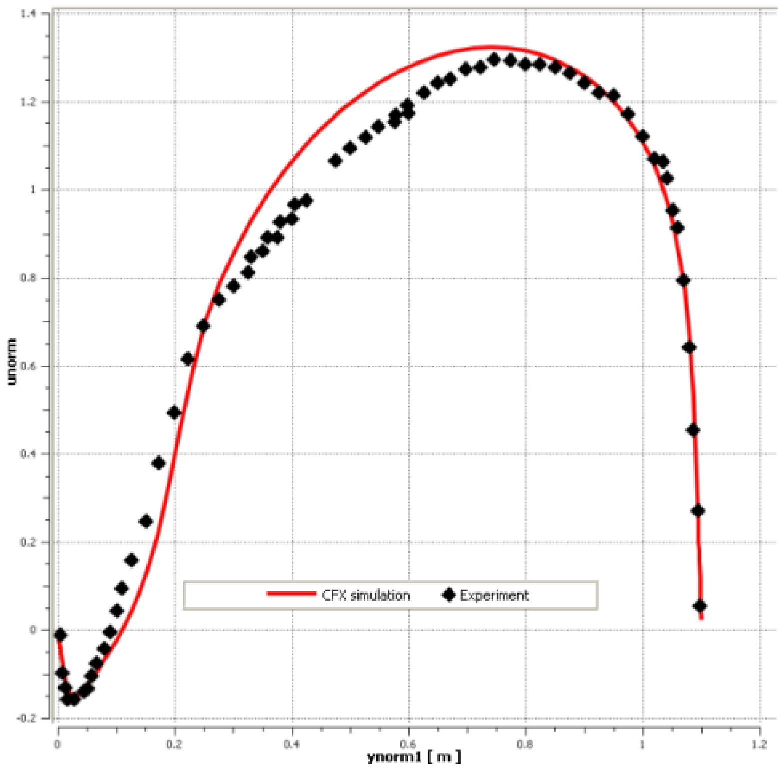


Figure 5 Comparison of Predicted Normalized X-Velocity along Transverse Direction at the Wave Trough



VMFL013: Turbulent Flow with Heat Transfer in a Backward-Facing Step

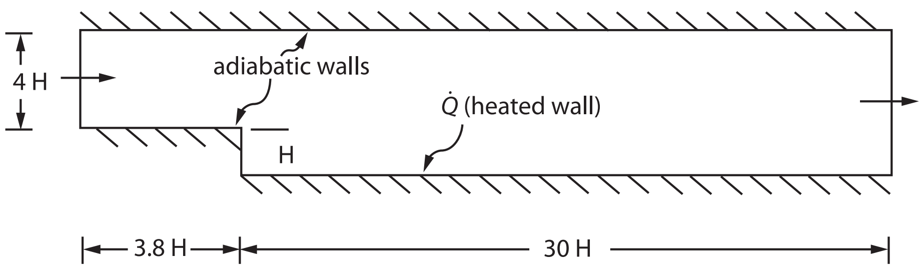
Overview

Reference	J.C.Vogel and J.K. Eaton.*"Combined Heat Transfer and Fluid Dynamic Measurements Downstream of a Backward-Facing Step". J. Heat Transfer*. Vol. 107. 922-929. 1985.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Incompressible, turbulent flow with heat convection and reattachment.
Input File	`step_ve.cas` for ANSYS FLUENT `VMFL013B_vv013.def` for ANSYS CFX

Test Case

The fluid flow and convective heat transfer over a 2–D backward-facing step is modeled. A constant heat-flux surface behind the sudden expansion leads to a separated and reattaching boundary layer that disturbs local heat transfer. Measured values of the distribution of the local Nusselt number along the heated wall are used to validate the CFD simulation.

Figure 1 Flow Domain



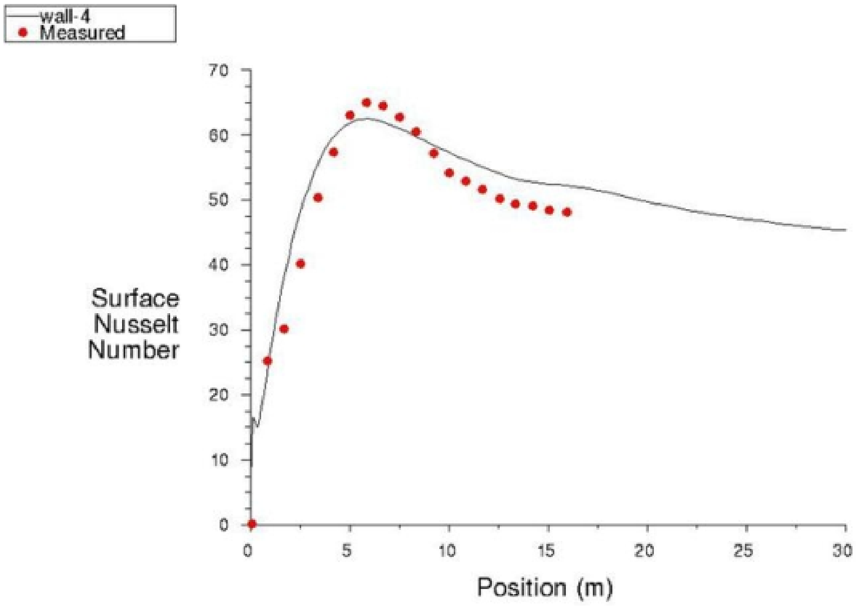
Material Properties for Dry Air	Geometry	Boundary Conditions
Density = 1 kg/m^3 Viscosity = 0.0001 kg/m-s Conductivity = 1.408 W/m-K Specific Heat = 10,000 J/kg-K	H = 1 m	Velocity profile at inlet corresponding to Re_H = 28,000 Wall heat transfer, $\dot{Q}$ = 1,000 W/m^2

Analysis Assumptions and Modeling Notes

A Cartesian non-uniform 121 x 61 mesh is used. The flow is steady and incompressible. Fluid properties are considered constant. Pressure based solver is used. The inlet boundary conditions are specified using the fully-developed profiles for the U-velocity, k, and epsilon. The incoming boundary layer thickness is 1.1 H. Under the given pressure conditions, the Reynolds number, Re_H is about 28,000 The RNG k-ε model with standard wall functions is used for accounting turbulence.

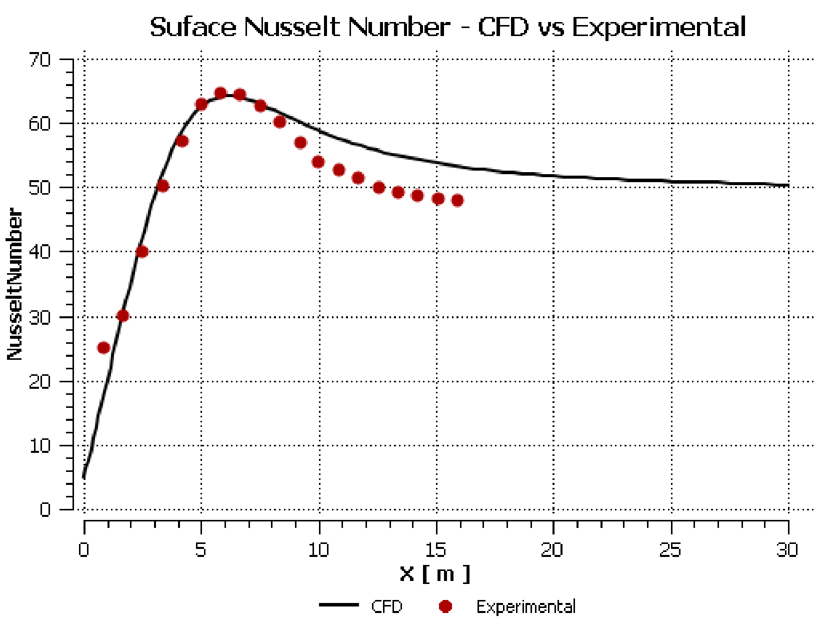
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Predicted Local Nusselt Number Distribution Along the Heated Wall with Experimental Data



Results Comparison for ANSYS CFX

Figure 3 Comparison of Predicted Local Nusselt Number Distribution Along the Heated Wall with Experimental Data



VMFL014: Species Mixing in Co-axial Turbulent Jets

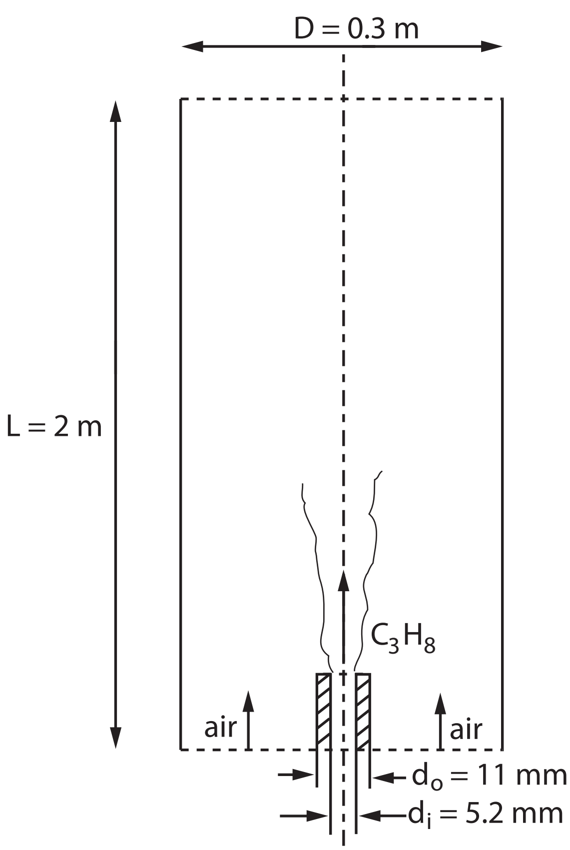
Overview

Reference	R.W. Schefer and R.W. Dibble. "Simultaneous Measurements of Velocity and Density in a Turbulent Non-premixed Flame". *AIAA Journal*, 23. 1070-1078. 1985. R.W. Schefer. "*Data Base for a Turbulent, Nonpremixed, Nonreacting Propane-Jet Flow*". http://www.sandia.gov/TNF/DataArch/ProJet.html
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Multi-Species flow, turbulent , jet mixing
Input File	**`san_jet.cas`** for ANSYS FLUENT **`VMFL014B_san_jet.def`** for ANSYS CFX

Test Case

A propane jet issues into a co-axial stream of air. There is turbulent mixing between the species in the axisymmetric tunnel. Only half of the domain is considered due to axial symmetry.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density: Incompressible ideal gas law Viscosity: $1.72X10^{-5}$ kg/m-s	Tunnel length = 2 m Tunnel diameter = 0.3 m Propane jet tube: Inner diameter = 5.2 mm Outer diameter = 11 mm	Inlet velocity of air = 9.2 m/s Inlet velocity of Propane – Specified as fully developed profile Inlet temperature (both streams) = 300 K Temperature at the wall = 300 K

Analysis Assumptions and Modeling Notes

The flow is steady. Species mixing is modeled with the three species; propane, oxygen, and nitrogen. There is no reaction.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Distribution of Propane Along Axis of the Jets

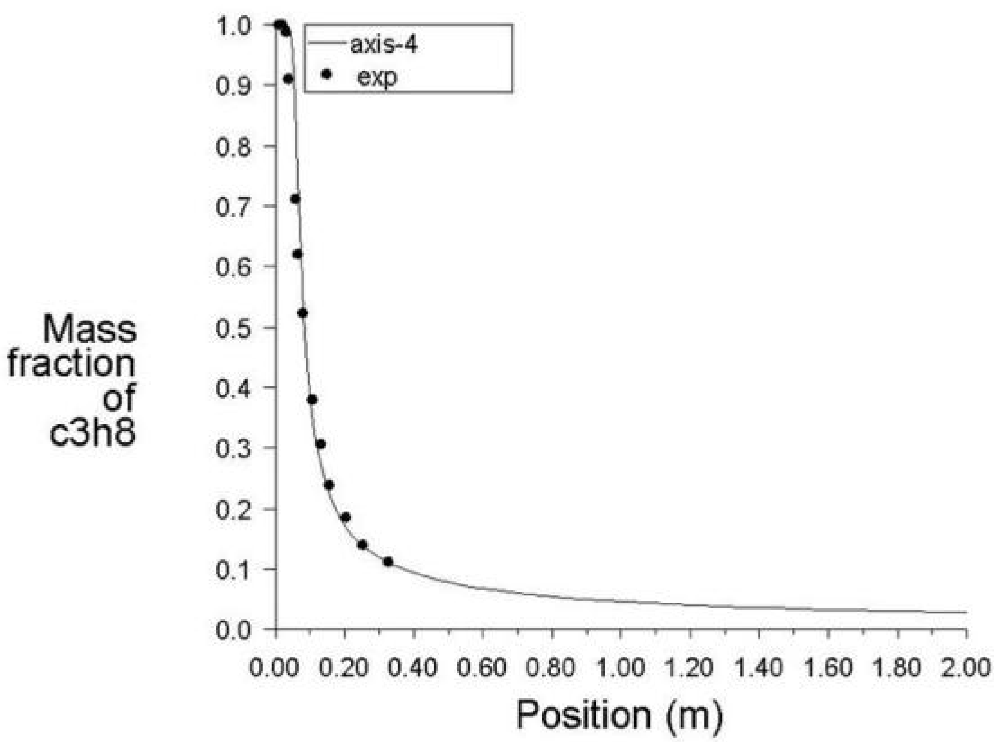
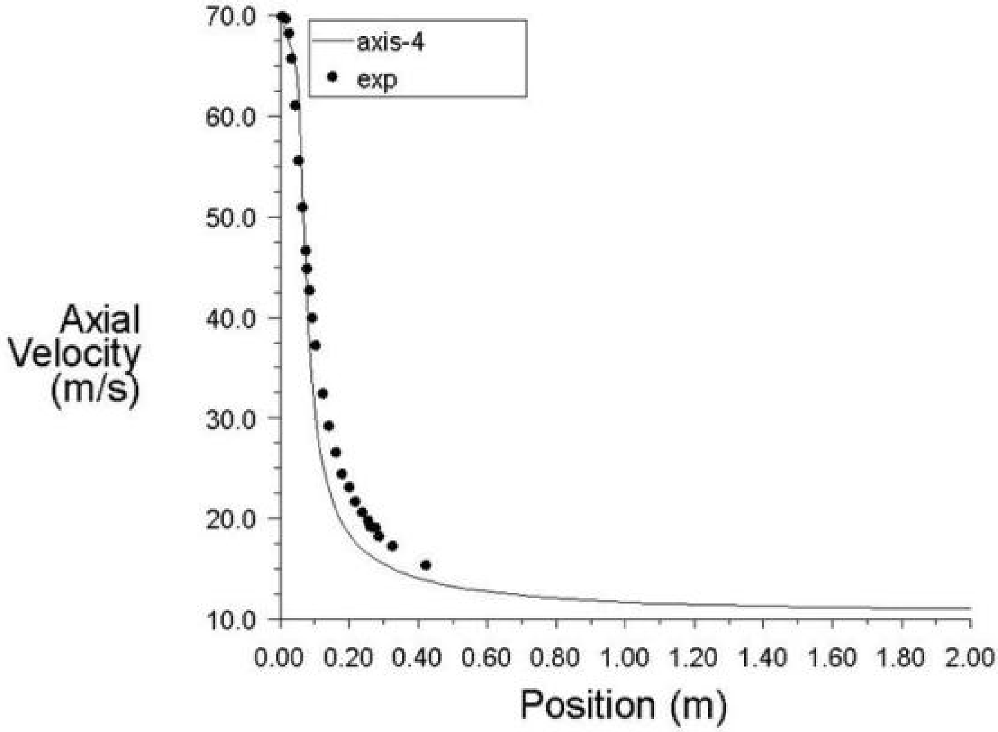


Figure 3 Comparison of Distribution of X-Velocity Along Axis of the Jets



Results Comparison for ANSYS CFX

Figure 4 Comparison of Distribution of Propane Along Axis of the Jets

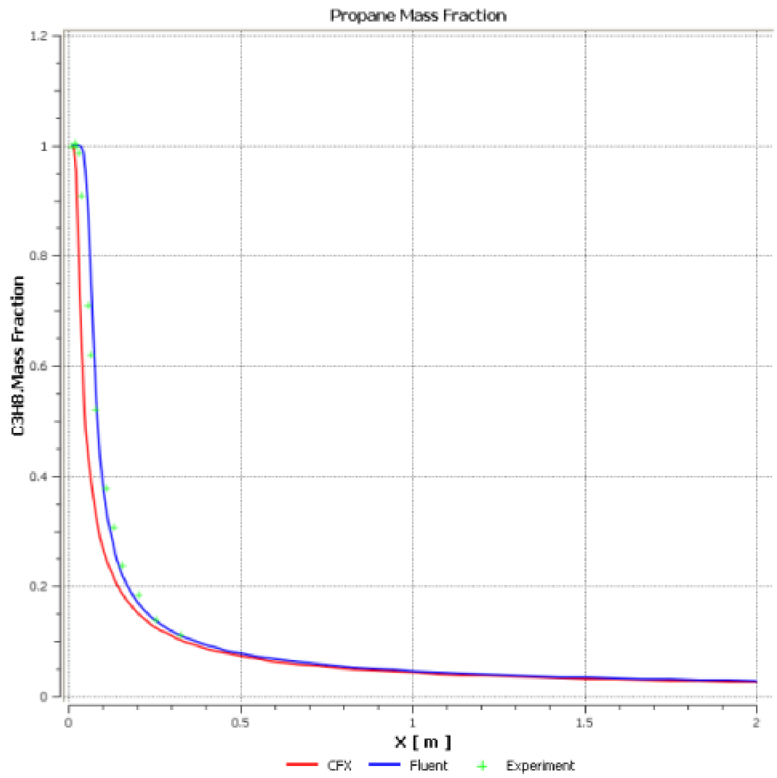
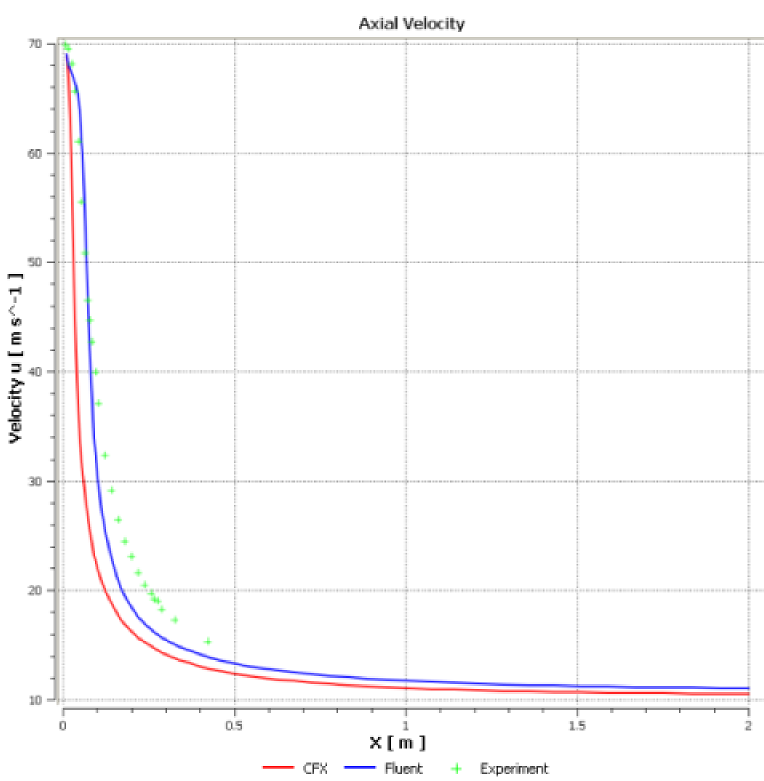


Figure 5 Comparison of Distribution of X-Velocity Along Axis of the Jets



VMFL015: Flow Through an Engine Inlet Valve

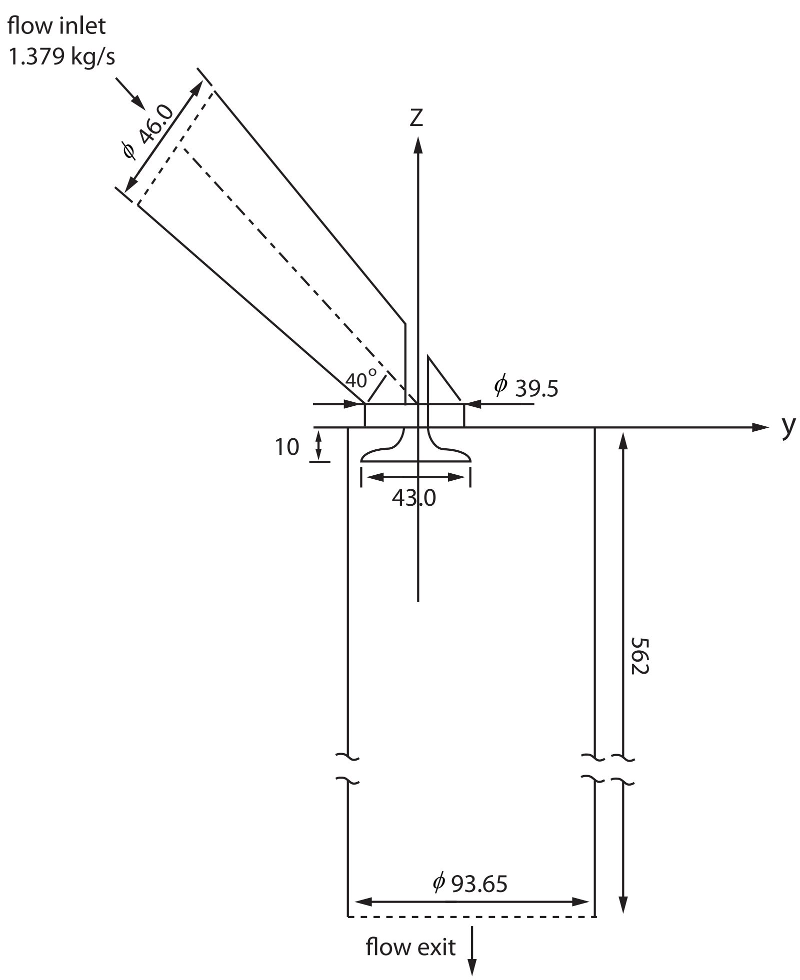
Overview

Reference	A. Chen, K.C. Lee, M. Yianneskis, and G. Ganti. "*Velocity Characteristics of Steady Flow Through a Straight Generic Inlet Port*". *International Journal for Numerical Methods in Fluids*, 21. 571-590. 1995.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	3–D turbulent flow
Input File	`valve10.cas` for ANSYS FLUENT `VMFL017B_VV017.def` for ANSYS CFX

Test Case

Flow in an idealized engine cylinder with a straight inlet port and a valve lift of 10 mm (the distance from the top of the cylinder to the bottom of the valve). The configuration of the inlet port, valve, and cylinder is shown in *Figure 1* (p. 55).

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density : 894 kg/m^3 Viscosity: 0.001529 kg/m-s	All dimensions shown in *Figure 1* (p. 55) are in mm.	Inlet velocity = 0.9282 m/s Inlet turbulent intensity = 10% Inlet turbulent length scale = 0.046m Outlet gauge pressure = 0 Pa

Analysis Assumptions and Modeling Notes

The flow is steady, isothermal and incompressible. The standard k-ε model with standard wall functions is used. The length of the cylinder is chosen to be large enough that it will not affect the flow in the cylinder.

Results Comparison for ANSYS FLUENT

Figure 2 (p. 56) and *Figure 3* (p. 57) compare ANSYS FLUENT's results with the experimental data (z-component of velocity at different heights).

Figure 2 Z-Velocity Component at Z= -5mm

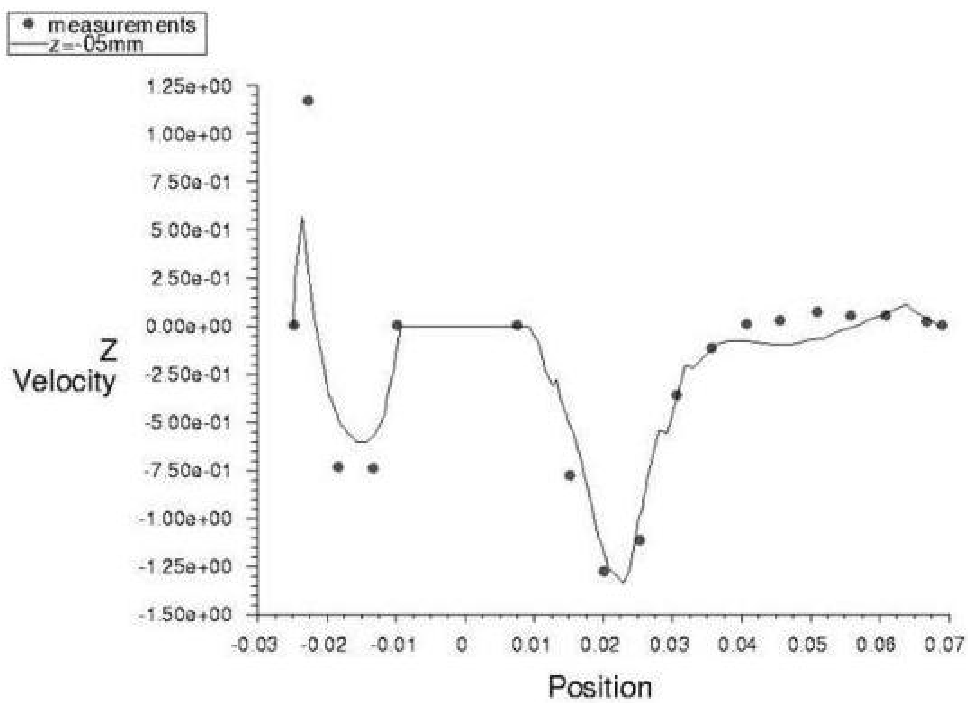
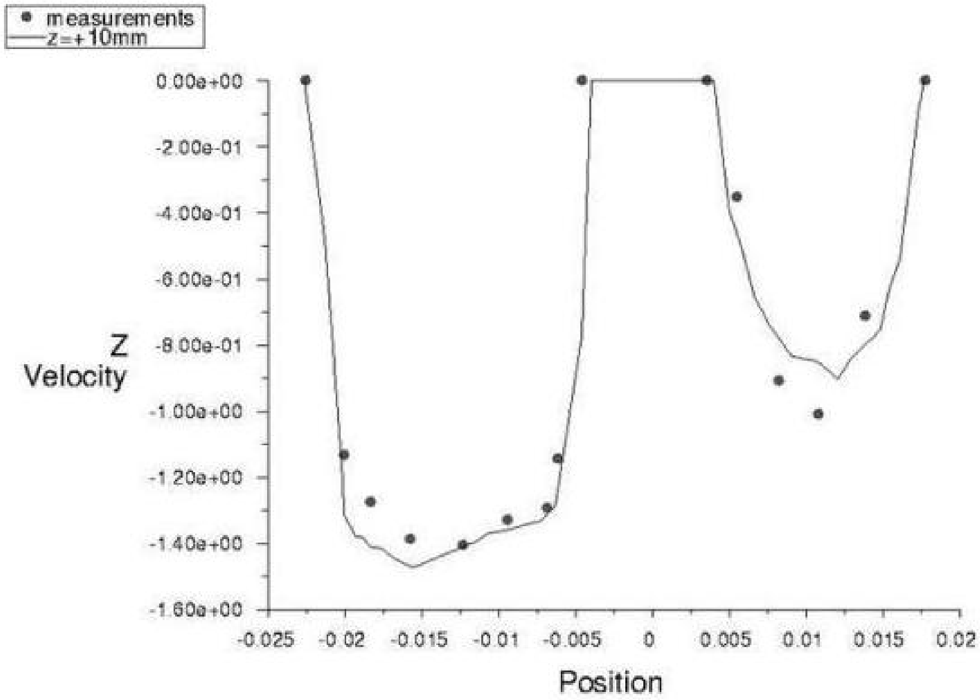


Figure 3 Z-Velocity Component at Z = +10mm



Results Comparison for ANSYS CFX

Figure 4 Z-Velocity Component at Z= -5mm

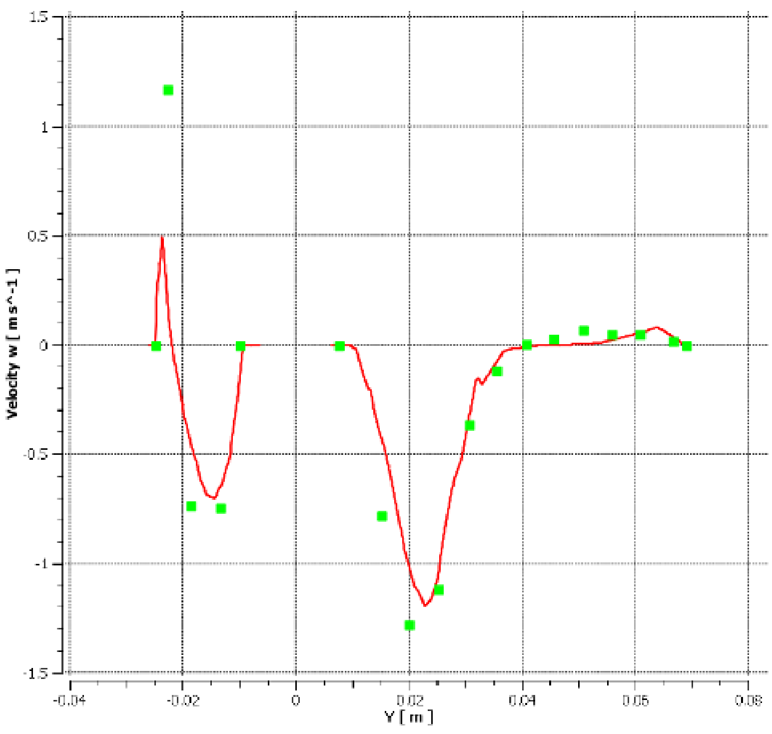
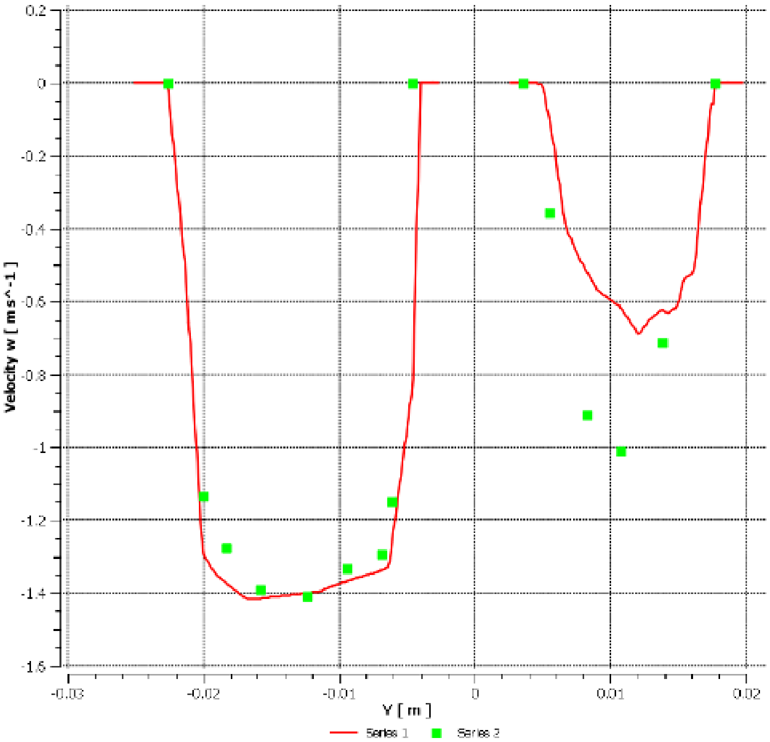


Figure 5 Z-Velocity Component at Z = +10mm



VMFL016: Turbulent Flow in a Transition Duct

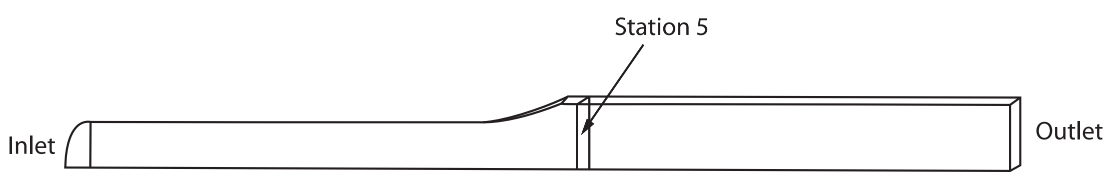
Overview

Reference	D.O. Davis and F.B. Gessner. "*Experimental Investigation of Turbulent Flow Through a Circular-to-Rectangular Transition Duct*". *AIAA Journal*, 30(2). 367-375. 1992.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	3–D Turbulent flow with separation, Reynolds stress model
Input Files	`tranduct-rsm-1.cas` for ANSYS FLUENT `transition_duct.def` for ANSYS CFX

Test Case

Turbulent flow through a circular-to-rectangular transition duct having the same inlet and outlet cross-sectional areas is modeled. The curvature of the duct walls induces a strong pressure-driven cross-flow that develops into a counter-rotating vortex pair near the short side walls of the duct. Due to symmetry of the flow field, only one fourth of the duct is modeled (as shown in *Figure 1* (p. 59)). Station 5 is located 23 m downstream of the inlet.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 5.13X10^{-6} kg/m-s	Inlet radius = 1 m Length of duct = 35 m	Inlet velocity: 1 m/s

Analysis Assumptions and Modeling Notes

The flow is steady. Reynolds Stress Model (RSM) is used to model turbulence.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Pressure Coefficient at Station 5

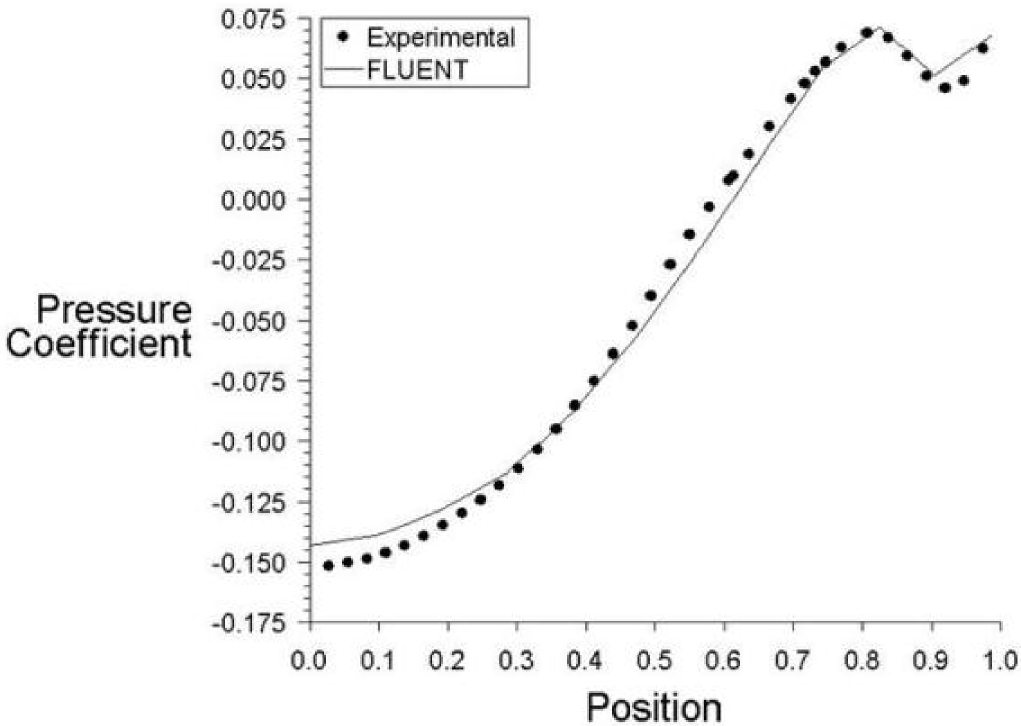
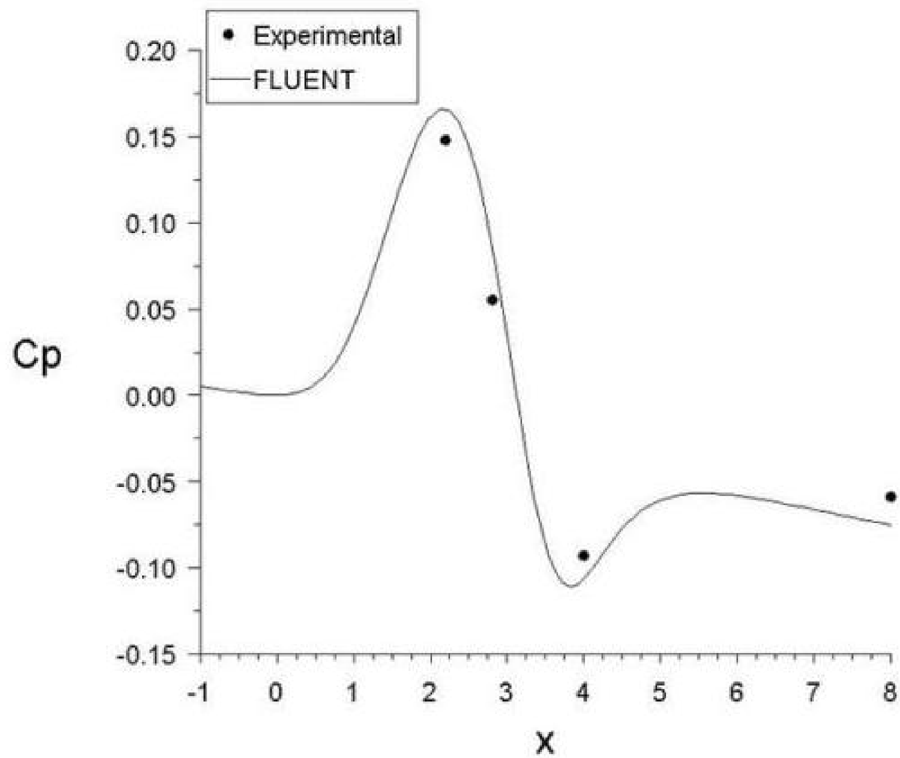
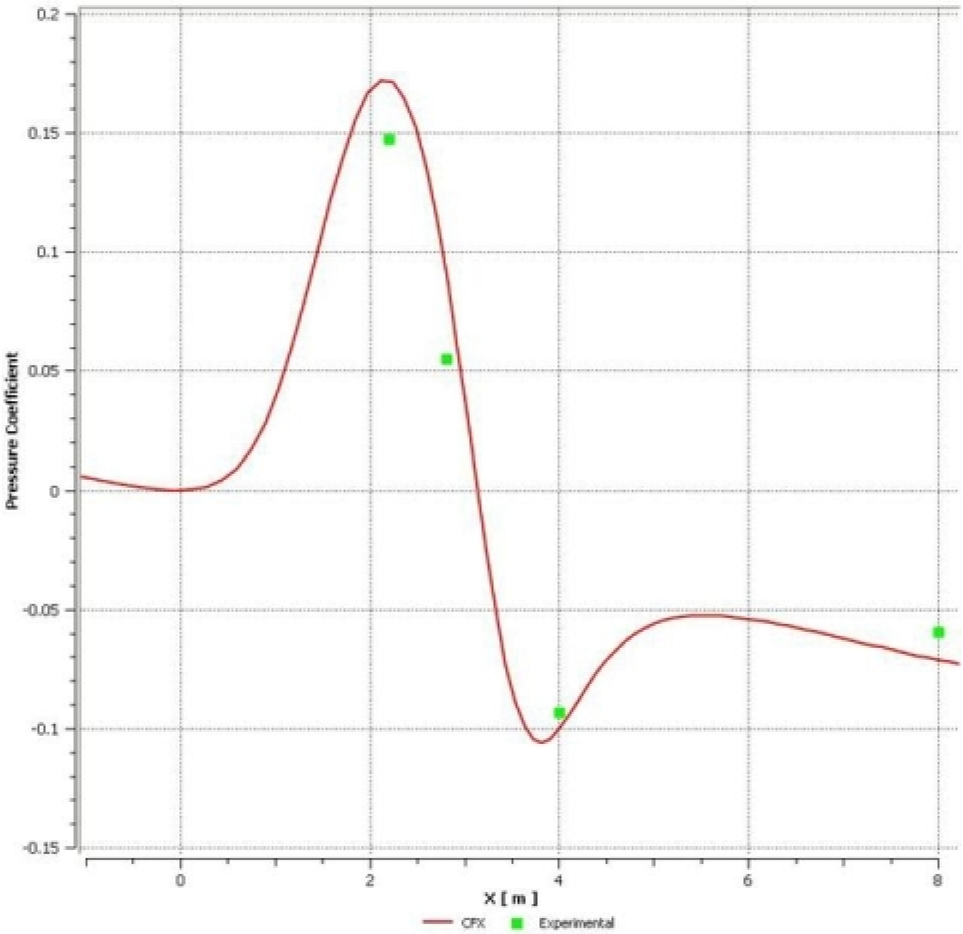


Figure 3 Comparison of Pressure Coefficient Along Centerline of the Duct



Results Comparison for ANSYS CFX

Figure 4 Comparison of Pressure Coefficient Along Centerline of the Duct



VMFL017: Transonic Flow over an RAE 2822 Airfoil

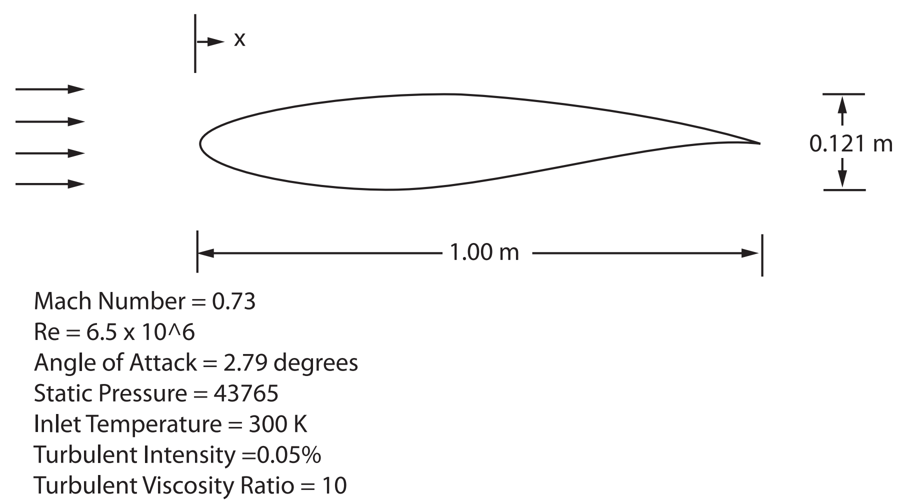
Overview

Reference	P.H. Cook, M.A. McDonald, and M.C.P. Firmin. "*AEROFOIL RAE 2822 Pressure Distribution and Boundary Layer and Wake Measurements*". *AGARD Advisory Report*. No. 138. 1979.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Compressible, turbulent flow
Input File	`r2822.cas` for ANSYS FLUENT `VMFL017B_VV017.def` for ANSYS CFX

Test Case

Flow over an RAE 2822 airfoil at a free-stream Mach number of 0.73. The angle of attack is 2.79°. The flow field is 2D, compressible (transonic), and turbulent. The geometry of the RAE 2822 airfoil is shown in *Figure 1* (p. 63). It is a thick airfoil with a chord length, c, of 1.00 m and a maximum thickness, d, of 0.121 m. The domain extends 55c from the airfoil, so that the presence of the airfoil is not felt at the outer boundary.

Figure 1 Geometry of the RAE 2822 Airfoil



Material Properties	Geometry	Boundary Conditions
Fluid: Air • Density: Ideal Gas • Viscosity: 1.983×10^{-5} kg/m-s • Thermal conductivity: 0.0242 W/m-K • Molecular Weight: 28.966 • Specific Heat: 1006.43 J/kg-K	Chord length = 1 m Maximum thickness = 0.121 m	The inlet conditions are: Mach number = 0.73 Re = 6.5×10^6 Static pressure = 43765 Pa Inlet temperature = 300 K

Material Properties	Geometry	Boundary Conditions
		Turbulent intensity = 0.05 % Turbulent viscosity ratio = 10

Analysis Assumptions and Modeling Notes

The implicit formulation of the density-based solver is used. The SST k-ω turbulence model is used to account for turbulence effects. The problem is solved in steady state mode.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Coefficients

Coefficients	Target	ANSYS FLUENT	Ratio
Drag	0.0168	0.0165	0.982
Lift	0.803	0.783	0.975

Results Comparison for ANSYS CFX

Table 2 Comparison of Coefficients

Coefficients	Target	ANSYS CFX	Ratio
Drag	0.0168	0.0165	0.982
Lift	0.803	0.7825	0.974

VMFL018: Shock Reflection in Supersonic Flow

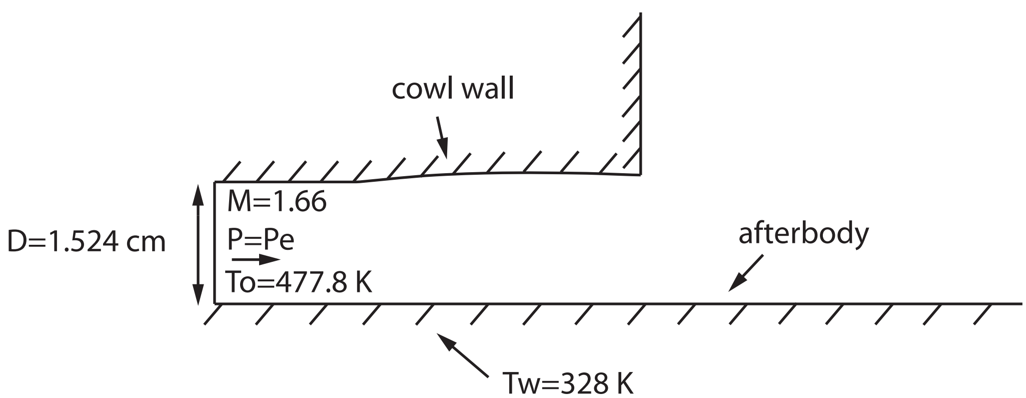
Overview

Reference	H. B. Hopkins, W. Konopka, and J. Leng. *Validation of scramjet exhaust simulation technique at Mach 6. NASA Contractor Report 3003*. 1979.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Reflecting shocks in supersonic flow; Compressible turbulent flow
Input File	`scram-nozzle-flow.cas` for ANSYS FLUENT `VMFL018B_VV018.def` for ANSYS CFX

Test Case

Supersonic flow from a nozzle that represents the exhaust nozzle of a supersonic combustion ramjet (SCRAMJET) is modeled. Jet from the nozzle is issued into a domain which is bounded on one side by an afterbody wall which is parallel to the centerline of the nozzle. Shocks propagating from the nozzle exit reflect from the afterbody. Measured values of (i) the distribution of wall pressure and (ii) heat transfer rate along the afterbody are used to validate the CFD simulation.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density: Ideal Gas Molecular Weight: 113.2 Viscosity: 1.7894 X 10^{-5} kg/m-s Thermal Conductivity: 0.0242 w/m-K Specific Heat: Temperature Dependent	D = 1.524 cm Length of cowl = 3.5 D	Inlet Total Pressure (gauge) = 551600 Pa Inlet Static Pressure (gauge) = 127100 Pa Inlet Total Temperature = 477.8 K Inlet Turbulent Intensity = 2 % Wall temperature = 328 K Outlet Pressure (gauge) = 2780 Pa

Analysis Assumptions and Modeling Notes

The flow is steady. Specific heat is defined as a linear function of temperature. Density based solver is used. Under the given pressure conditions, the inlet Mach number is about 1.66.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Predicted Static Pressure Distribution on the Afterbody with Experimental Data

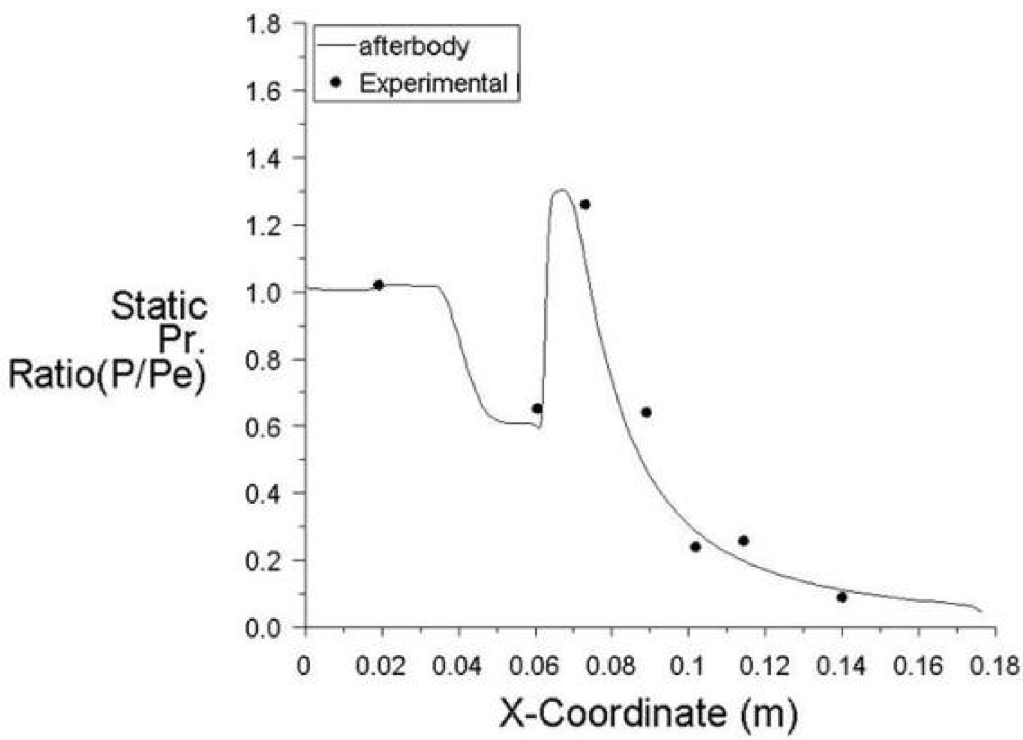
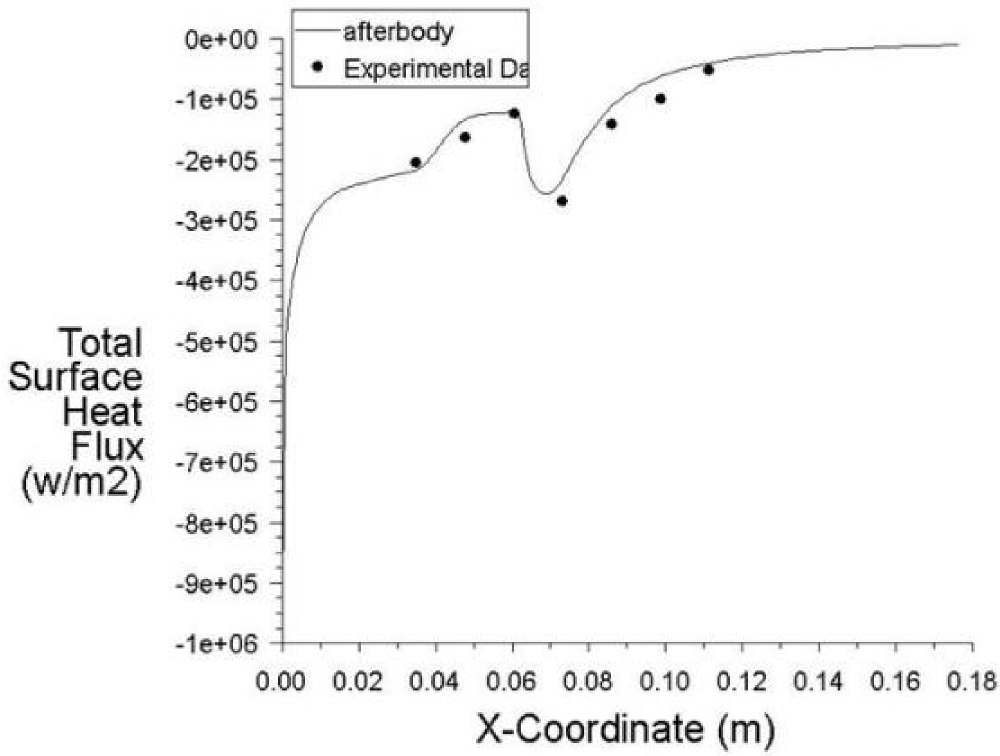


Figure 3 Comparison of Predicted Total Heat Flux Along the Afterbody with Experimental Data



Results Comparison for ANSYS CFX

Figure 4 Comparison of Predicted Static Pressure Distribution on the Afterbody with Experimental Data

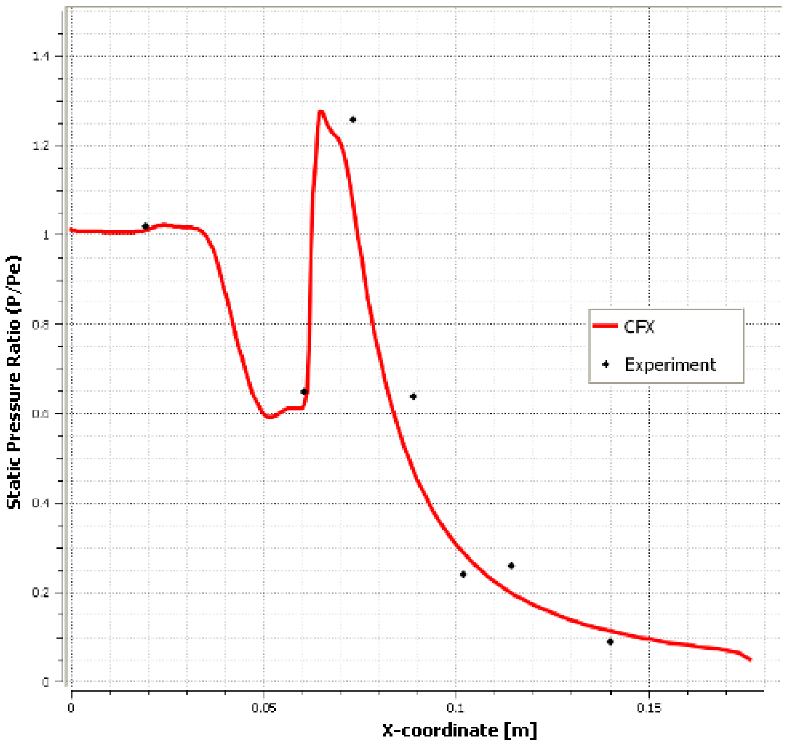
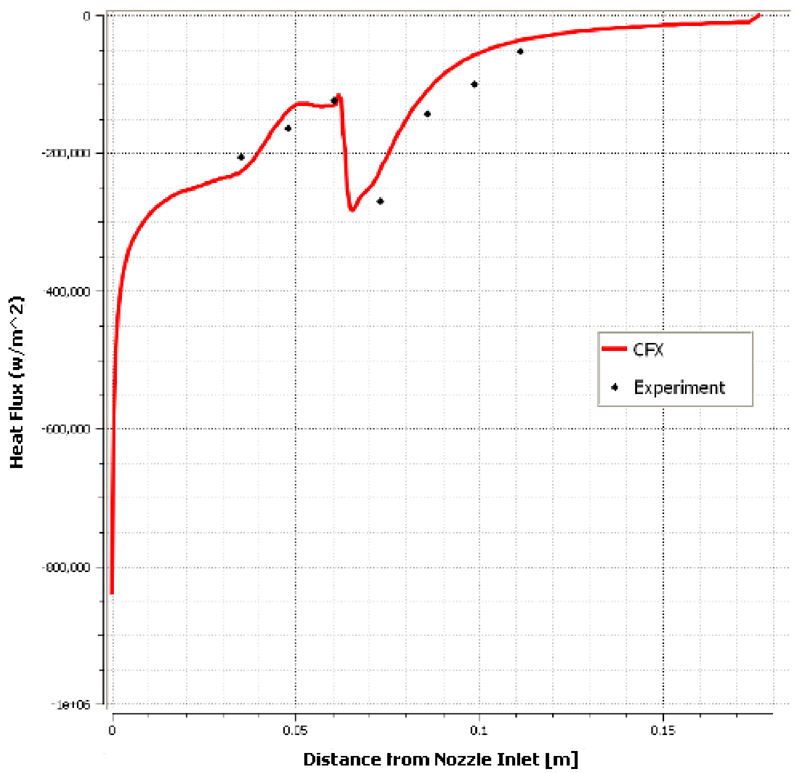


Figure 5 Comparison of Predicted Total Heat Flux Along the Afterbody with Experimental Data



VMFL019: Transient Flow near a Wall Set in Motion

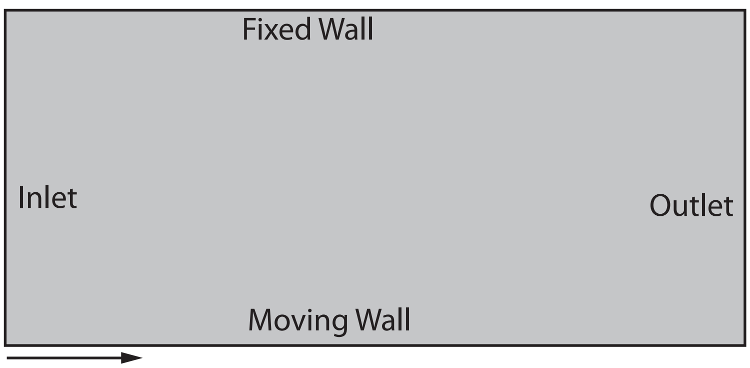
Overview

Reference	Boundary Layer Theory, H. Schlichting & K. Gersten, 8th Edition, 1999; Page 126-127.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Unsteady flow, moving wall
Input File	`VMFL019_FLUENT.cas` for ANSYS FLUENT `VMFL019_CFX.def` for ANSYS CFX

Test Case

Flow near a wall suddenly set into motion is modeled. The start up flow is modeled as a transient problem with a constant wall-velocity at t (time) > 0. The flow is highly viscous and the velocity is 0 at t= 0.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density = 1000 kg/m^3 Viscosity = 1 kg/m-s	Dimensions of the domain: 0.75 m X 0.3 m	Velocity of the moving wall = 0.02 m/s Gauge Pressure at Inlet = 0 N/m^2 Gauge Pressure at Outlet = 0 N/m^2

Analysis Assumptions and Modeling Notes

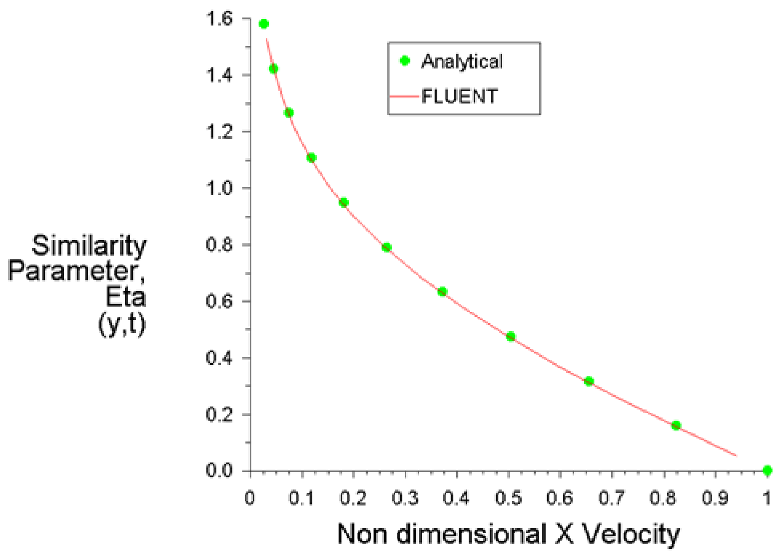
The density based solver is used in ANSYS FLUENT. Pressure boundaries are specified to model the driving head in the direction of flow. The fluid is at rest initially (t = 0). The similarity parameter is defined as:

$$\eta(y,t) = y / \left(2\sqrt{\nu t}\right)$$

Where ν is the kinematic viscosity.

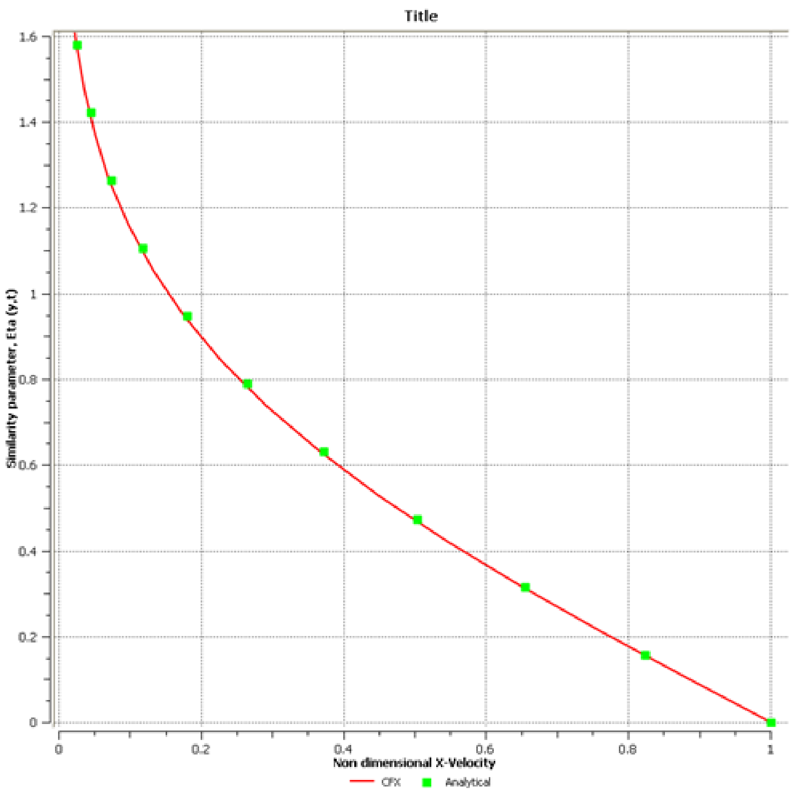
Results Comparison using ANSYS FLUENT

Figure 2 Comparison of Velocity Profile Near the Wall at Outlet



Results Comparison using ANSYS CFX

Figure 3 Comparison of Velocity Profile Near the Wall at Outlet



VMFL020: Adiabatic Compression of Air in Cylinder by a Reciprocating Piston

Overview

Reference	L. D. Russell and G. A. Adebiyi. "*Classical Thermodynamics*". Saunders College Publishing. Philadelphia, PA (Now Oxford University Press). 1993.
Solver	ANSYS FLUENT (ANSYS CFX simulation is not available for this case)
Physics/Models	Dynamic Mesh, Transient flow with ideal gas effects
Input File	`box2d_remesh.cas`

Test Case

Air undergoes adiabatic compression due to the movement of a piston inside a rectangular box, representing a cylinder geometry in 2–D as shown in *Figure 1* (p. 75). The Top Dead Center (TDC) corresponds to a crank angle of 360°. The piston moves back after reaching TDC.

Figure 1 In-Cylinder Piston Description

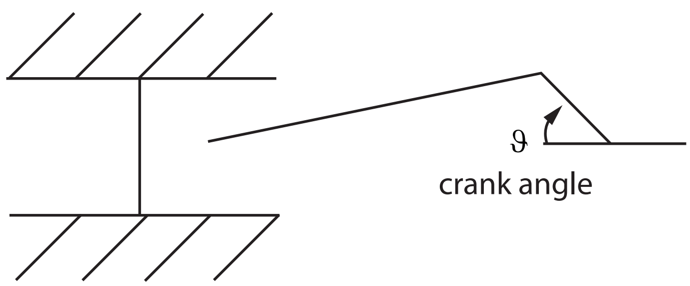
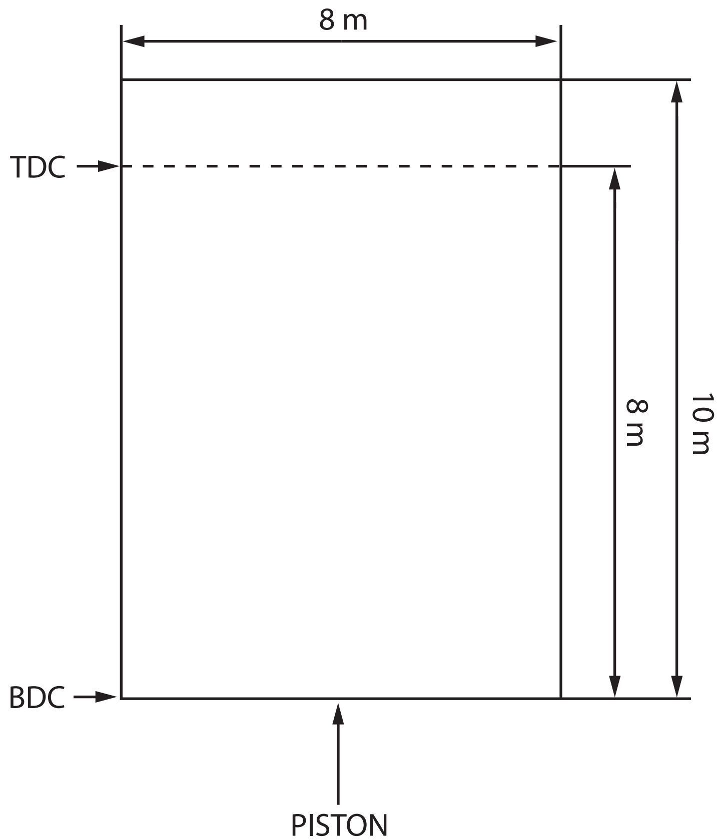


Figure 2 Flow Domain



Material Properties	Geometry	Boundary Conditions
Ideal gas law for density Viscosity = 1.7894 X 10^{-5} kg/m-s	Length of the block = 10 m Width of the block = 8 m	Movement of the piston modeled using deforming mesh

Analysis Assumptions and Modeling Notes

The compression within the cylinder is assumed to be adiabatic. The Spring-based smoothing method with local remeshing is used for modeling the dynamic mesh motion.

Results Comparison

Figure 3 Comparison of Static Temperature Variation with Time

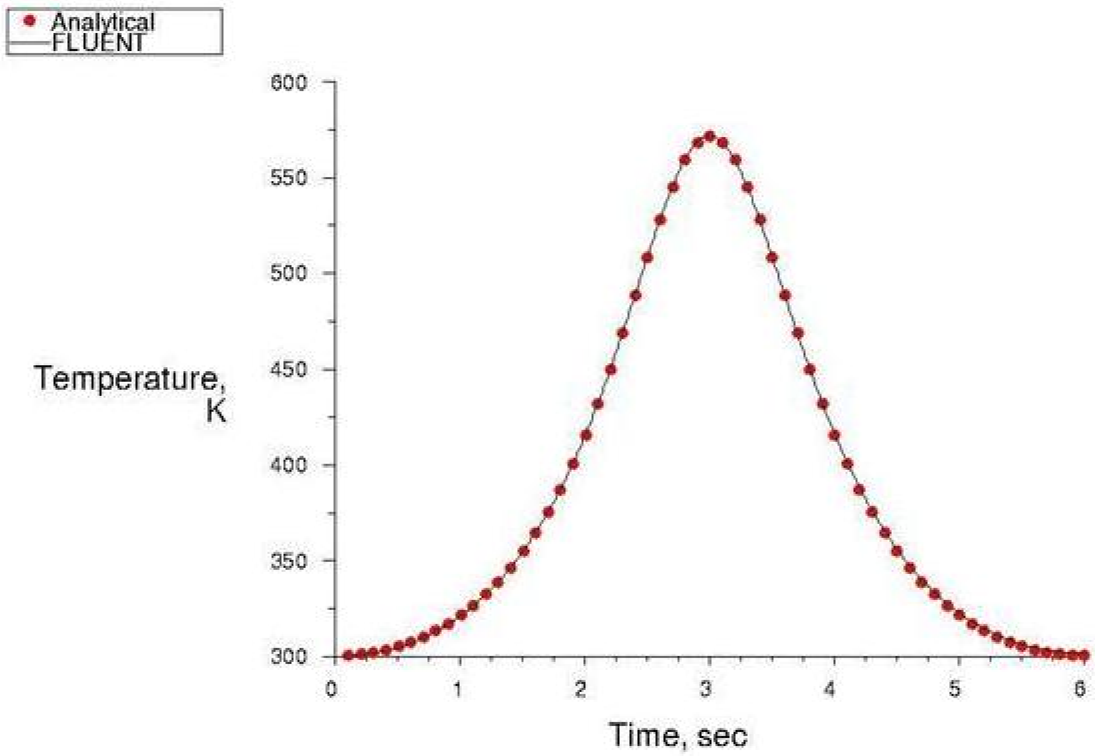
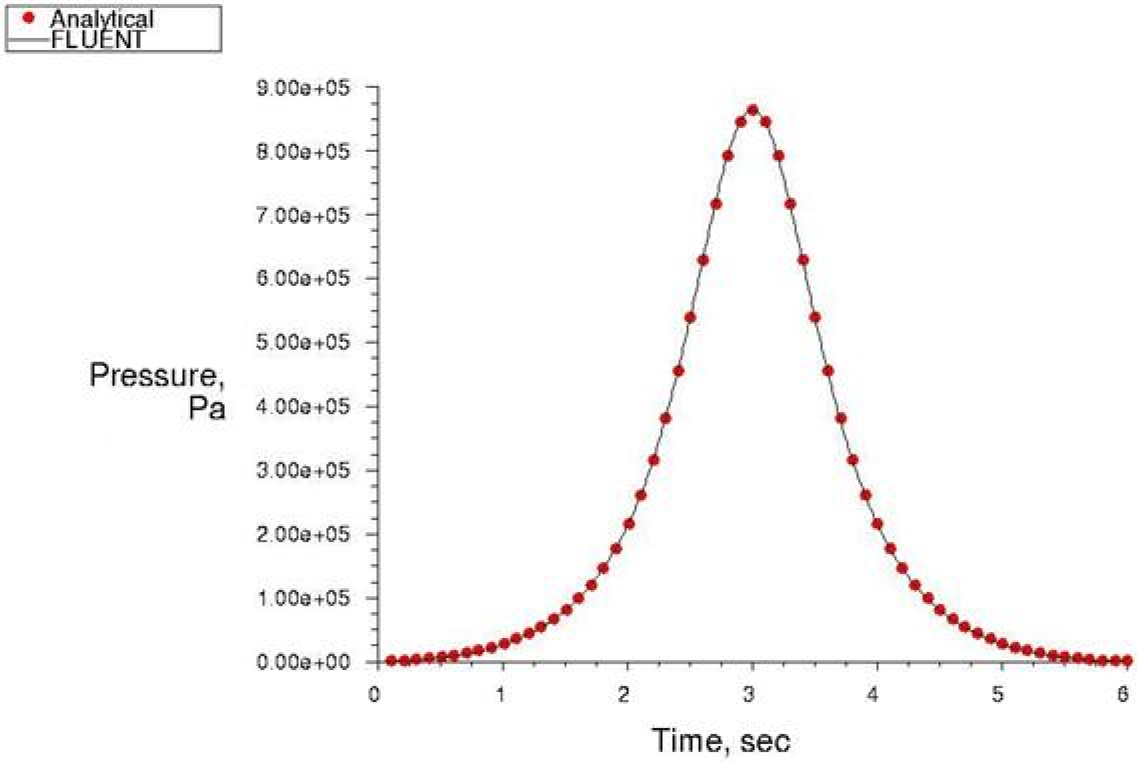


Figure 4 Comparison of Static Pressure Variation with Time



VMFL021: Cavitation over a Sharp-Edged Orifice Case A: High Inlet Pressure

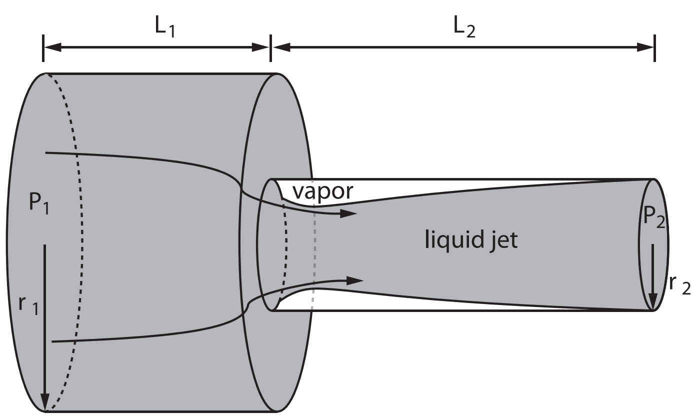
Overview

Reference	W.H. Nurick. "*Orifice Cavitation and Its Effects on Spray Mixing*". Journal of Fluids Eng.. Vol.98. 681-687. 1976.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent multiphase flow with cavitation and phase change
Input File	`cav_orifice_HP.cas` for ANSYS FLUENT `VMFL021B_VV021.def` for ANSYS CFX

Test Case

A steady, axisymmetric, multiphase (water/steam) flow, with phase change taking place. Due to sudden contraction a low pressure region occurs near the sharp edge which results in cavitation. *Figure 1* (p. 79)depicts the orifice geometry. Flow direction is from left to right.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Liquid: Water Density : 1000 kg/m^3 Viscosity: 0.001 kg/m-s **Gas: Water-Vapor** Density: 0.02558 kg/m^3 Viscosity: $1.26x10^{-6}$ kg/m-s	L_1 = 1.60 cm L_2 = 3.20 cm r_1 = 1.15 cm r_2 = 0.40 cm	P_1 = 250,000,000 Pa P_2 = 95,000 Pa T = 300 K P_{vapor} = 3,540 Pa

Analysis Assumptions and Modeling Notes

The flow is steady and incompressible. Pressure based solver is used. Standard k-ε model with standard wall functions is used for turbulence. The Zwart-Gerber-Belamri cavitation model is applied together with mixture multiphase model.

For analysis of results, we calculate and compare the discharge coefficient with the experimental data.

The coefficient of discharge, C_d, is the ratio of the mass flow rate through the nozzle to the theoretical maximum mass flow rate:

$$C_d = \frac{\dot{m}}{A\sqrt{2\rho(P_1 - P_2)}}$$

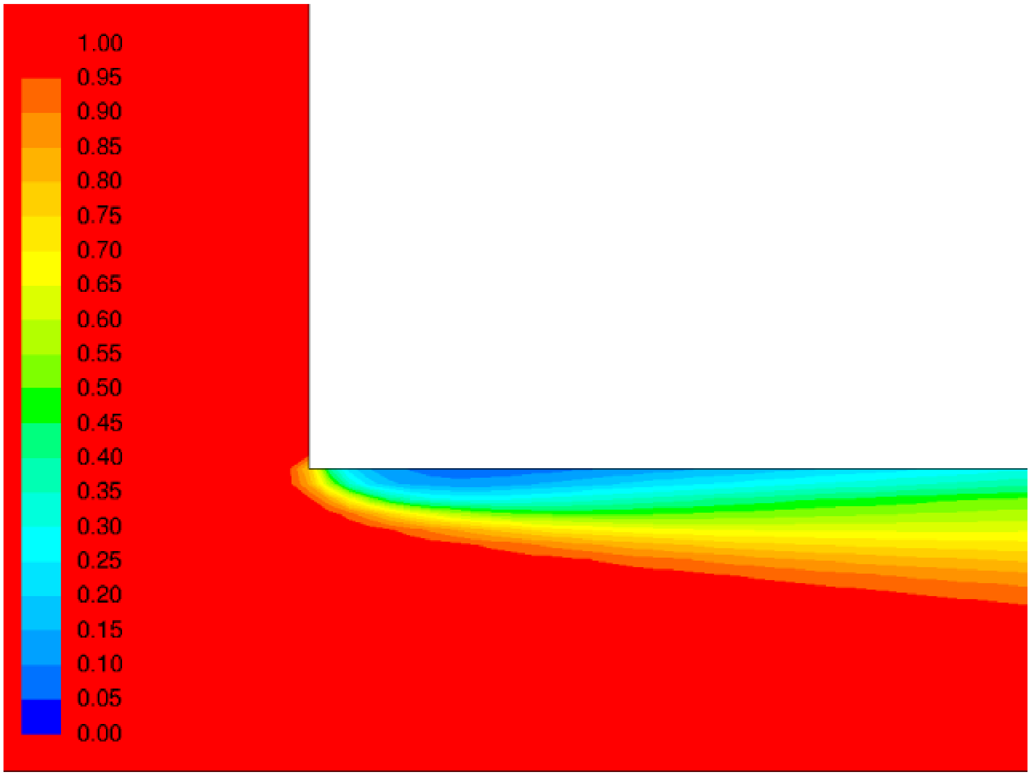
In the above equation, $\dot{m}$ is the mass flow rate as calculated by the CFD solver.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Discharge Coefficient

	Target	ANSYS FLUENT	Ratio
Coefficient of Discharge	0.620	0.631	1.018

Figure 2 Contours of Liquid (Water) Volume Fraction



Results Comparison for ANSYS CFX

Table 2 Comparison of Discharge Coefficient

	Target	ANSYS CFX	Ratio
Coefficient of Discharge	0.620	0.6429	1.037

VMFL022: Cavitation over a Sharp-Edged Orifice Case B: Low Inlet Pressure

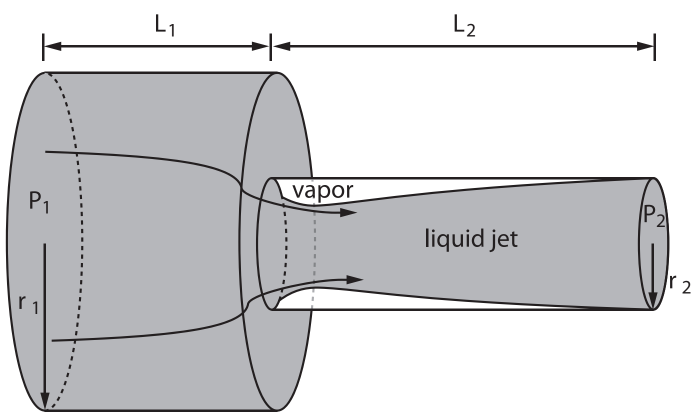
Overview

Reference	W.H. Nurick. "*Orifice Cavitation and Its Effects on Spray Mixing*". *Journal of Fluids Eng.*. Vol.98. 681-687. 1976.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent multiphase flow with cavitation and phase change
Input File	**`cav_orifice_LP.cas`** for ANSYS FLUENT **`VMFL022B_VV022.def`** for ANSYS CFX

Test Case

A steady, axisymmetric, multiphase (water/steam) flow, with phase change taking place. Due to sudden contraction a low pressure region occurs near the sharp edge which results in a weak cavitation. *Figure 1* (p. 83) depicts the orifice geometry. Flow direction is from left to right.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Liquid: Water Density: 1000 kg/m^3 Viscosity: 0.001 kg/m-s **Gas: Water-Vapor** Density: 0.02558 kg/m^3 Viscosity: 1.26x10^{-6} kg/m-s	L_1 = 1.60 cm L_2 = 3.20 cm r_1 = 1.15 cm r_2 = 0.40 cm	P_1 = 250,000 Pa P_2 = 95,000 Pa T = 300 K P_{vapor} = 3,540 Pa

Analysis Assumptions and Modeling Notes

The flow is steady and incompressible. Pressure based solver is used. Standard k-ε model with standard wall functions is used for turbulence. The Zwart-Gerber-Belamri cavitation model is applied together with mixture multiphase model.

For analysis of results, we calculate and compare the discharge coefficient with the experimental data.

The coefficient of discharge, C_d, is the ratio of the mass flow rate through the nozzle to the theoretical maximum mass flow rate:

$$C_d = \frac{\dot{m}}{A\sqrt{2\rho(P_1 - P_2)}}$$

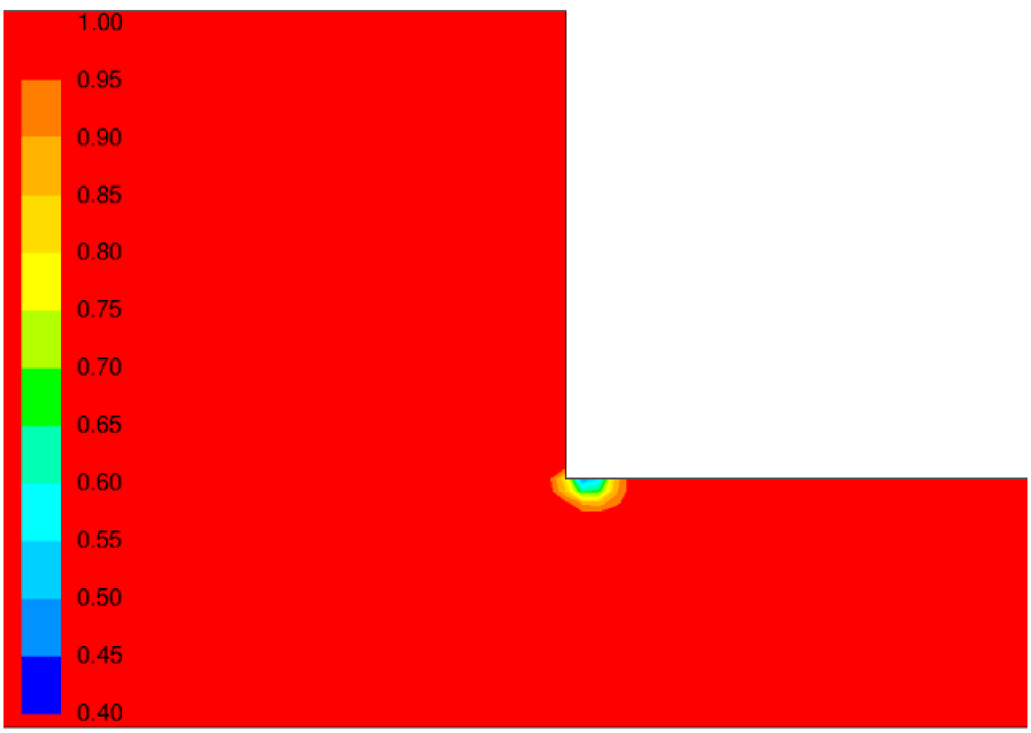
In the above equation, $\dot{m}$ is the mass flow rate as calculated by the CFD solver.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Discharge Coefficient

	Target	ANSYS FLUENT	Ratio
Coefficient of Discharge	0.780	0.777	0.996

Figure 2 Contours of Liquid (Water) Volume Fraction



Results Comparison for ANSYS CFX

Table 2 Comparison of Discharge Coefficient

	Target	ANSYS CFX	Ratio
Coefficient of Discharge	0.780	0.8051	1.032

VMFL023: Oscillating Laminar Flow Around a Circular Cylinder

Overview

Reference	F. M. White. "*Fluid Mechanics*". 3rd Edition. McGraw-Hill Book Co.. New York, NY. 1994. S.J. Kim and C.M. Lee. "*Numerical Investigation of Cross-Flow Around a Circular Cylinder at a Low-Reynolds Number Flow Under an Electromagnetic Force*". KSME *International Journal*. Vol 16. No. 3. 363-375. 2002.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Laminar, transient flow
Input File	**`cyl_2d.cas`** for ANSYS FLUENT **`VMFL023B_osc_cyl.def`** for ANSYS CFX

Test Case

The purpose of this case is to validate the ability of ANSYS FLUENT and ANSYS CFX to predict the flow structure as well as the reattachment length and Strouhal number against experimental results. The present calculations are confined to the low-Reynolds-number regime (Re = 100), which encompasses unsteady asymmetric flow.

Figure 1 Flow Domain

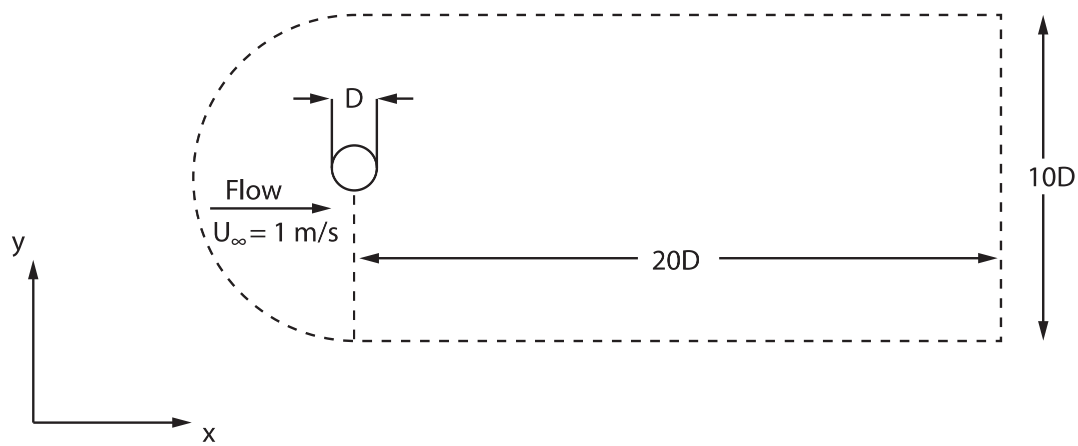


Table 1 Materials, Geometry, and Boundary Conditions

Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 0.02 kg/m-s	Diameter of the cylinder = 2 m	U_{∞} = 1 m/s

Analysis Assumptions and Modeling Notes

The flow is laminar, and unsteady. An FFT analysis of the lift coefficient on the cylinder wall is presented to determine the frequency of oscillations. The Strouhal number corresponding to the maximum magnitude of oscillations is presented in the Table below.

Results Comparison for ANSYS FLUENT

The formula for the Strouhal number is S = (N * D)/U_∞, where N is the frequency, D is the diameter of the cylinder, and U_∞ is the freestream velocity.

Table 2 Predicted Strouhal Number for Re = 100

	Target	ANSYS FLUENT	Ratio
Strouhal Number	0.165	0.173	1.048

Results Comparison for ANSYS CFX

Table 3 Predicted Strouhal Number for Re = 100

	Target	ANSYS CFX	Ratio
Strouhal Number	0.165	0.172	1.040

VMFL024: Interface of Two Immiscible Liquids in a Rotating Cylinder

Overview

Reference	T. Sugimoto and M. Iguchi. "*Behavior of Immiscible Two Liquid Layers Contained in Cylindrical Vessel Suddenly Set in Rotation*". *ISIJ Int.*, 42. 338-343. 2002.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Multiphase (Volume of Fluid), transient flow, body force
Input File	`rot-cyl_2liq_vof.cas` for ANSYS FLUENT `VMFL024B_rot_cyl.def` for ANSYS CFX

Test Case

Laminar interface between two immiscible liquids, water and silicon oil, inside a vertical cylinder which is set in rotation starting from a state of rest. The silicone oil layer rests on top of the water due to its lower density. The cylinder is sealed at the top. The vessel is set to rotate with a constant angular velocity.

Figure 1 Flow Domain

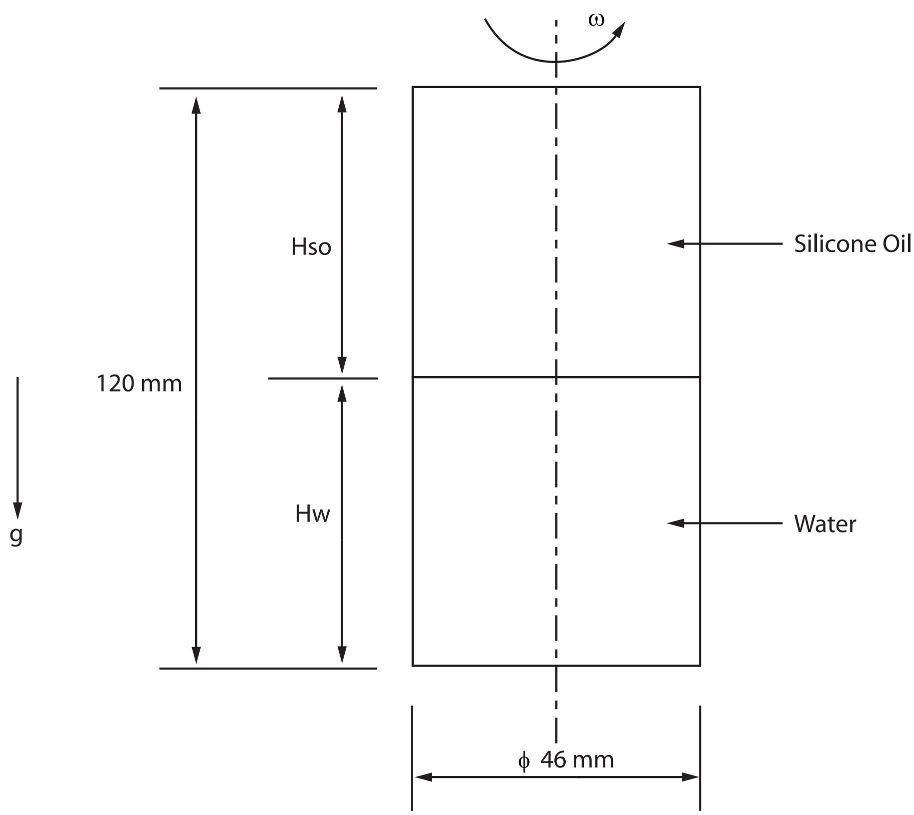


Table 1 Materials, Geometry, and Boundary Conditions

Material Properties	Geometry	Boundary Conditions
Water :	Diameter of the cylinder = 46mm	All walls are set up at rotational speed of 2.39577 rad/s

Material Properties	Geometry	Boundary Conditions
Density: 1030 kg/m^3 Viscosity: 0.0103 kg/m-s **Silicon Oil:** Density: 935 kg/m^3 Viscosity: 0.00935 kg/m-s	Height of the cylinder = 120 mm	

Analysis Assumptions and Modeling Notes

The flow is laminar, unsteady and axisymmetric. Non-dimensionalized swirl velocity, defined as **Swirl velocity / (Rotational speed X Cylinder radius)** is used to validate the results.

Results Comparison for ANSYS FLUENT

Table 2 Comparison of the Non-Dimensional Swirl Velocity at Various Radial Locations (for a Given Axial Location, X = 20mm) at Time t = 80 s

Radial locations (at x = 20 mm)	Target	ANSYS FLUENT	Ratio
4.83 mm	0.21	0.2093	0.997
9.43 mm	0.41	0.4109	1.002
14.26 mm	0.62	0.6221	1.003

Results Comparison for ANSYS CFX

Table 3 Comparison of the Non-Dimensional Swirl Velocity at Various Radial Locations (for a Given Axial Location, X = 20mm) at Time t = 80 s

Radial locations (at x = 20 mm)	Target	ANSYS CFX	Ratio
4.83 mm	0.21	0.2093	0.997
9.43 mm	0.41	0.4109	1.002
14.26 mm	0.62	0.6221	1.003

VMFL025: Turbulent Non-Premixed Methane Combustion with Swirling Air

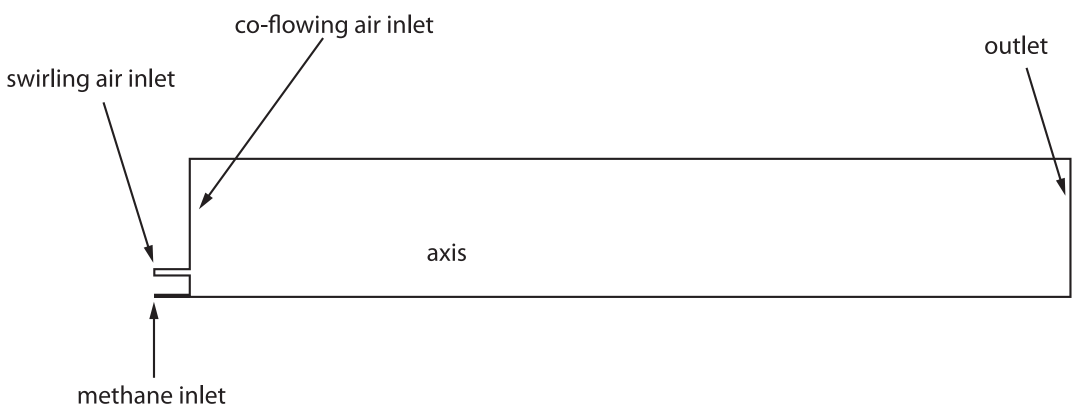
Overview

Reference	P.A.M. Kalt, Y.M. Al-Abdeli, A.R. Masri, and R.S. Barlow . "*Swirling turbulent non-premixed flames of methane: Flow field and compositional structure*". *Proc. Combust. Inst.*, 29. 1913-1919. 2002. Y.M. Al-Abdeli and A.R. Masri. "*Stability Characteristics and Flow Fields of Turbulent Swirling Jet Flows*". *Combust. Theory and Modeling*, 7. 731-766. 2003.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent swirling flow with reaction, non-premixed combustion model, Radiation heat transfer, Discrete ordinates method
Input File	`non-premix_17k-final.cas` for ANSYS FLUENT `VMFL025B_CFX` for ANSYS CFX

Test Case

Air and Methane enter as separate streams into an annular chamber. Air issues as a swirling jet and also as a separate co-flowing stream with axial velocity. Both the air streams are free of methane. Species mixing and combustion take place in the axisymmetric chamber. Radiative heat transfer is taken into account.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Species mixture properties specified through PDF file Viscosity: 1.72 x 10^{-05} kg/m-s Refractive Index = 1	The fuel (methane only) inlet has a diameter of 3.6 mm. The air inlet for the annular shroud has an inner diameter of 50mm and an outer diameter of 60 mm. Co-flowi ng air inlet has an outer diameter of 310 mm.	Methane inlet velocity: 32.7 m/s Axial velocity of swirling air: 38.2 m/s Swirl velocity of air: 19.1 m/s Co-flowing air velocity: 20 m/s

Material Properties	Geometry	Boundary Conditions
		Walls are adiabatic

Analysis Assumptions and Modeling Notes

The flow is steady. Realizable k-ε is used to model turbulence. Discrete ordinates method used to model radiation. The walls are treated as adiabatic. Non-premixed combustion model is used to model reactions.

Results Comparison ANSYS FLUENT

Figure 2 Comparison of Axial Velocity at X = 40mm

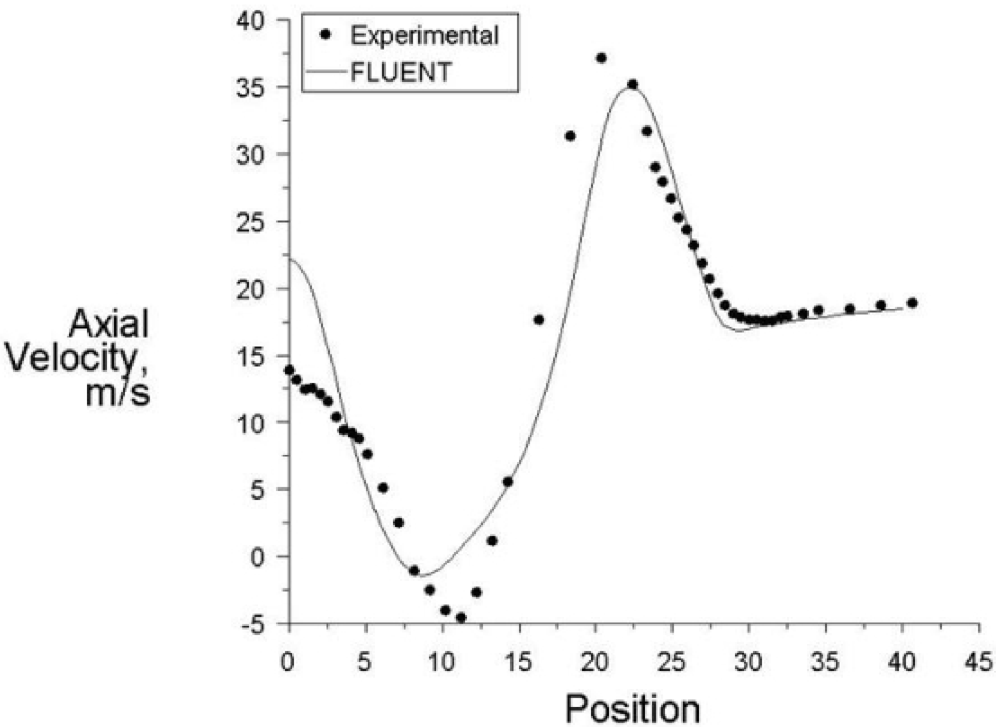


Figure 3 Comparison of Swirl Velocity at X = 40mm

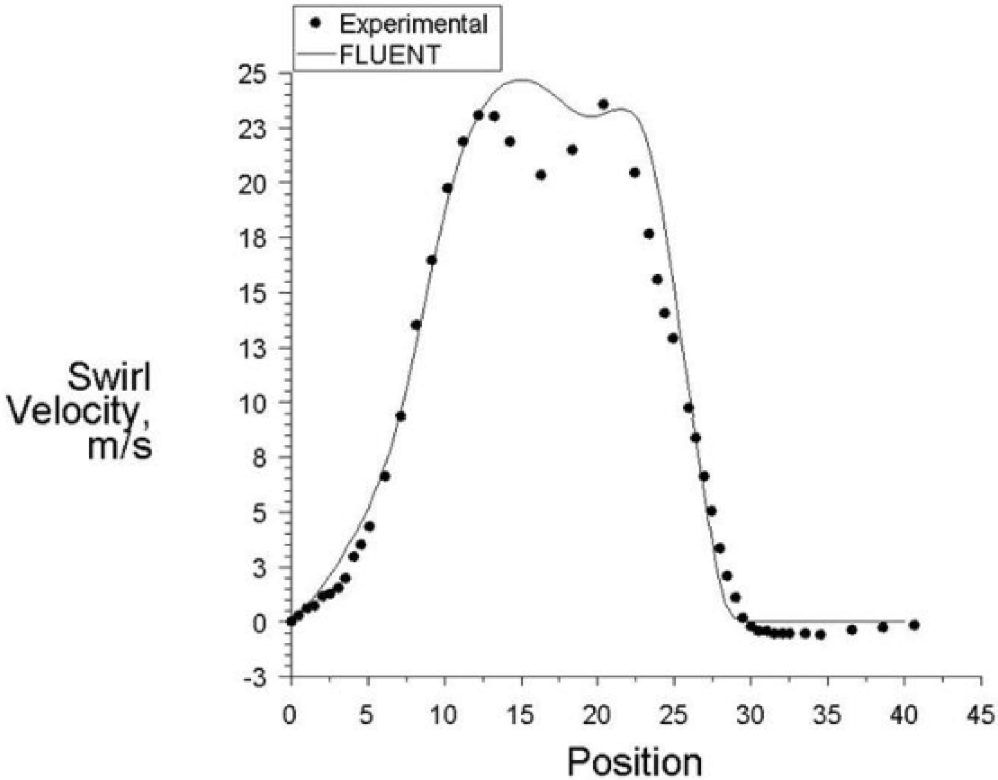


Figure 4 Comparison of Temperature at X = 40mm

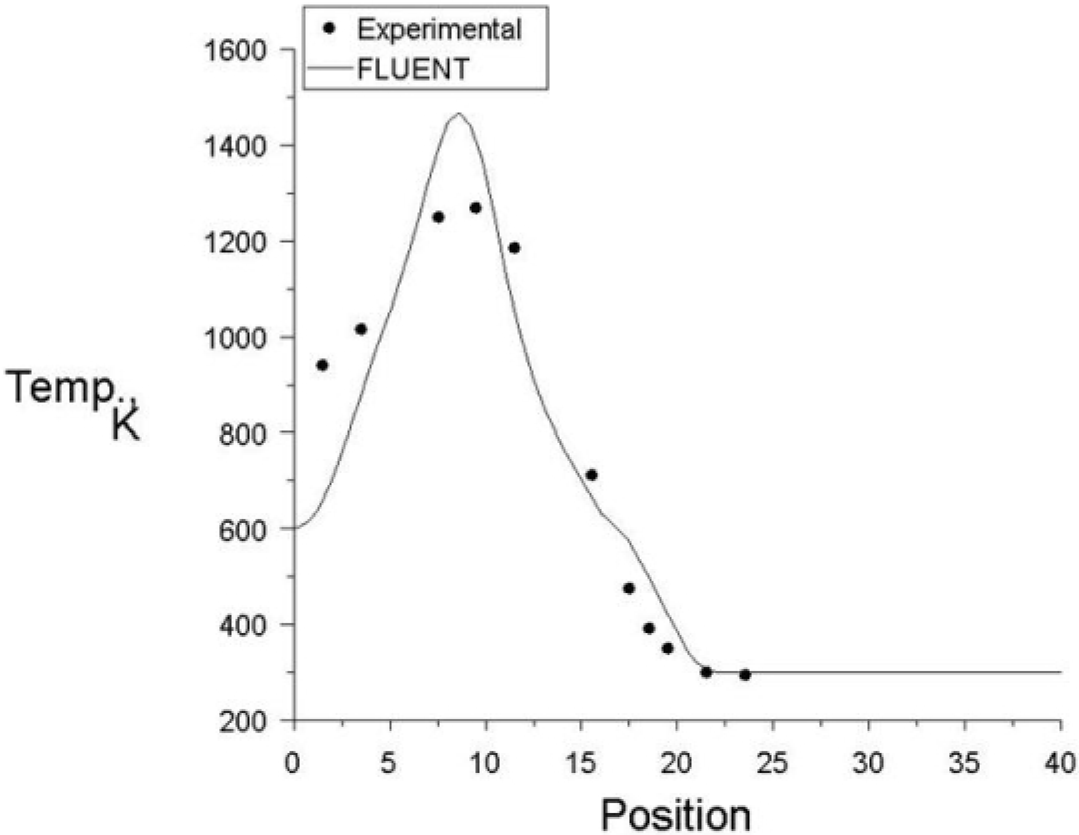
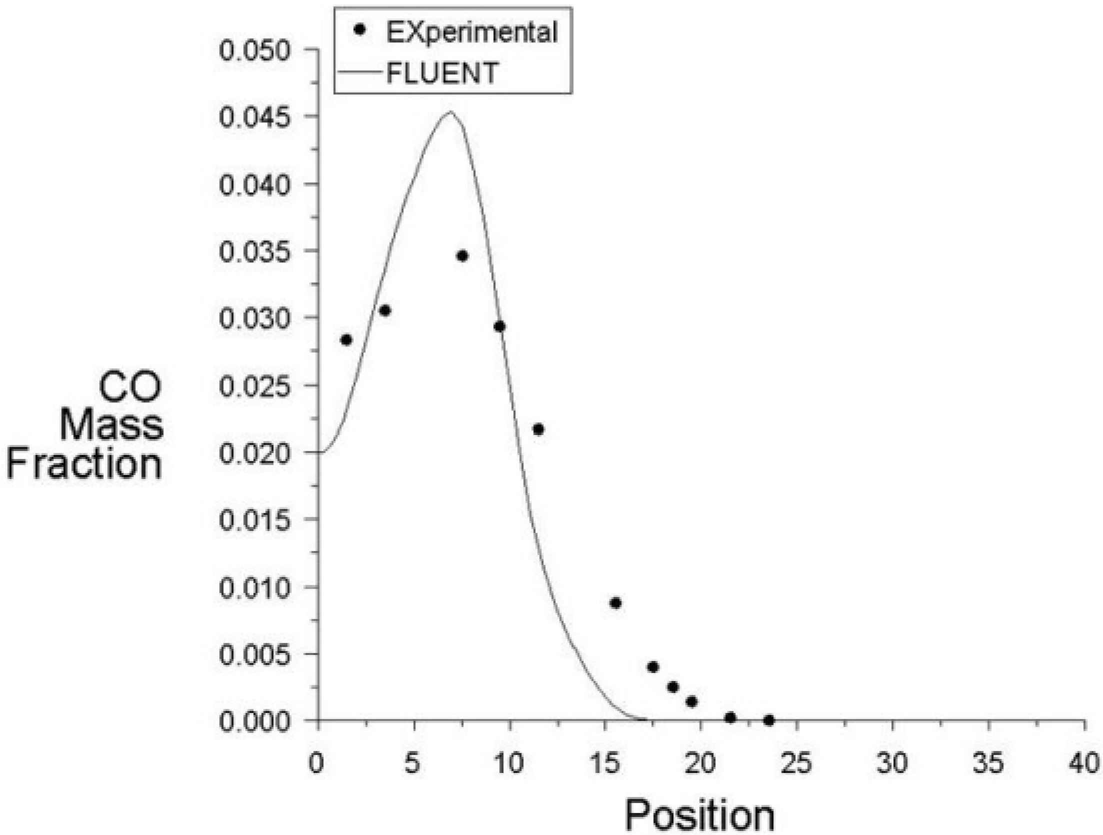


Figure 5 Comparison of Mass Fraction of CO at X = 40mm



Results Comparison for ANSYS CFX

Figure 6 Comparison of Axial Velocity at X = 40mm

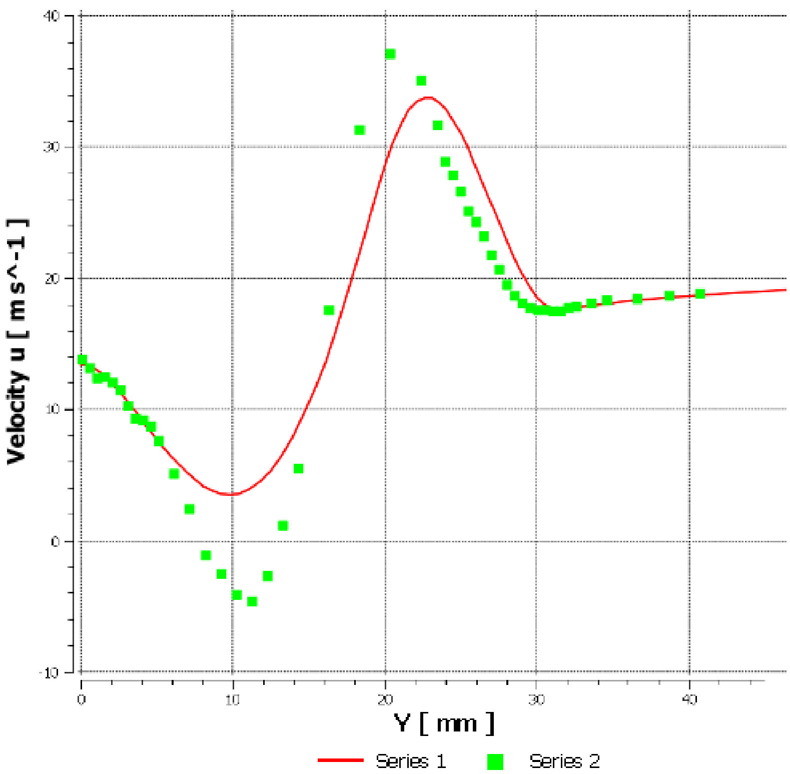


Figure 7 Comparison of Swirl Velocity at X = 40mm

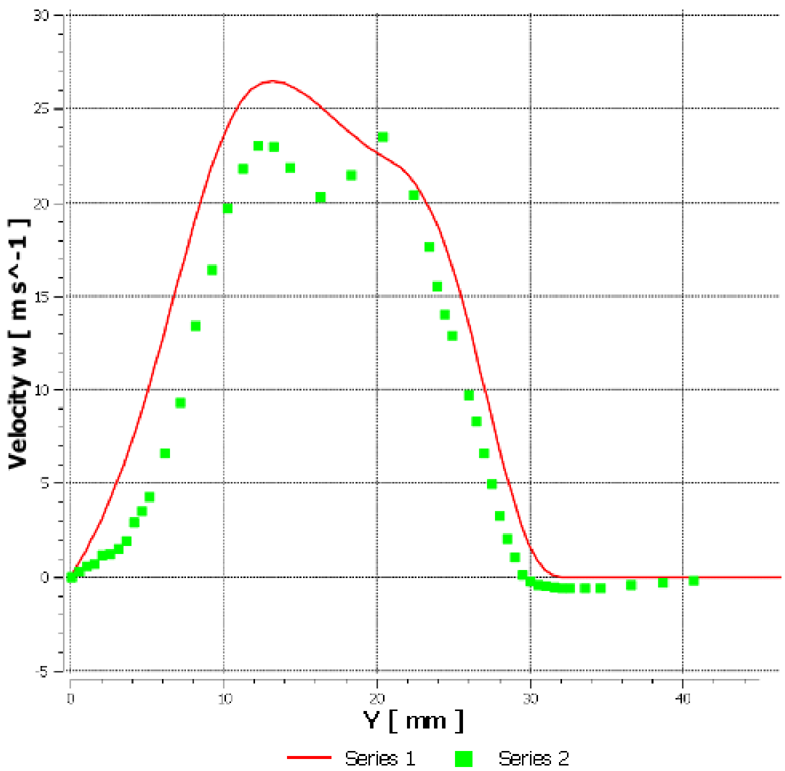


Figure 8 Comparison of Temperature at X = 40mm

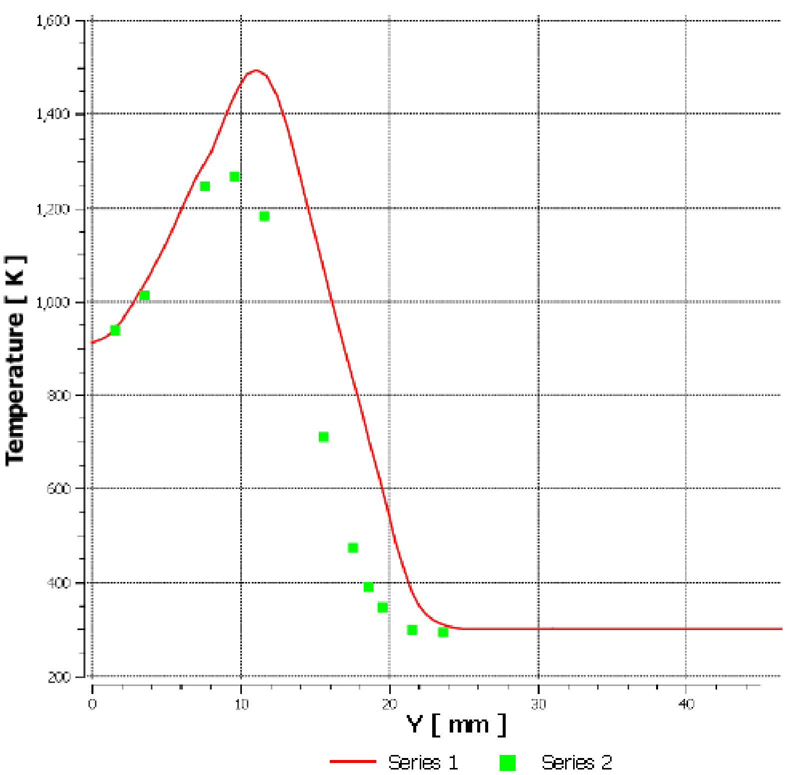
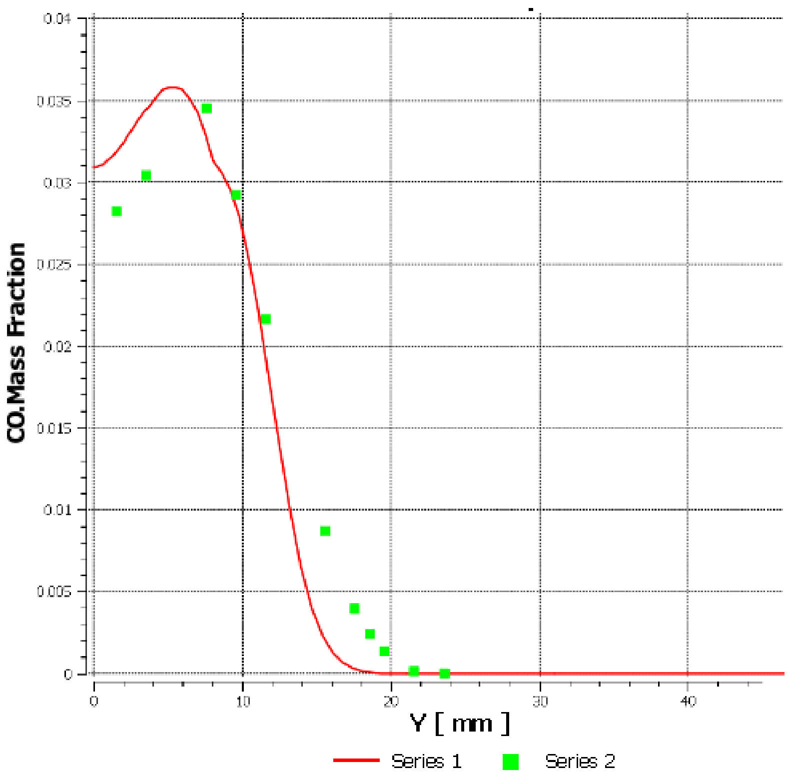


Figure 9 Comparison of Mass Fraction of CO at X = 40mm



VMFL026: Supersonic Flow with Real Gas Effects inside a Shock Tube

Overview

Reference	K. Mohamed and M. Paraschivoiu. "*Real Gas Numerical Simulation of Hydrogen Flow*". *2nd International Energy Conversion Engineering Conference*. Providence, Rhode Island. Aug. 16-19, 2004.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Transient Compressible flow, Real Gas effects, Shock
Input File	`realgas_shock-tube.cas` for ANSYS FLUENT `VMFL026B_CFX.def` for ANSYS CFX

Test Case

Transient flow inside a hydrogen filled shock tube is modeled. A diaphragm separating regions of high and low pressures ruptures at t = 0 thereby creating a shock wave in the tube.

Figure 1 Flow Domain

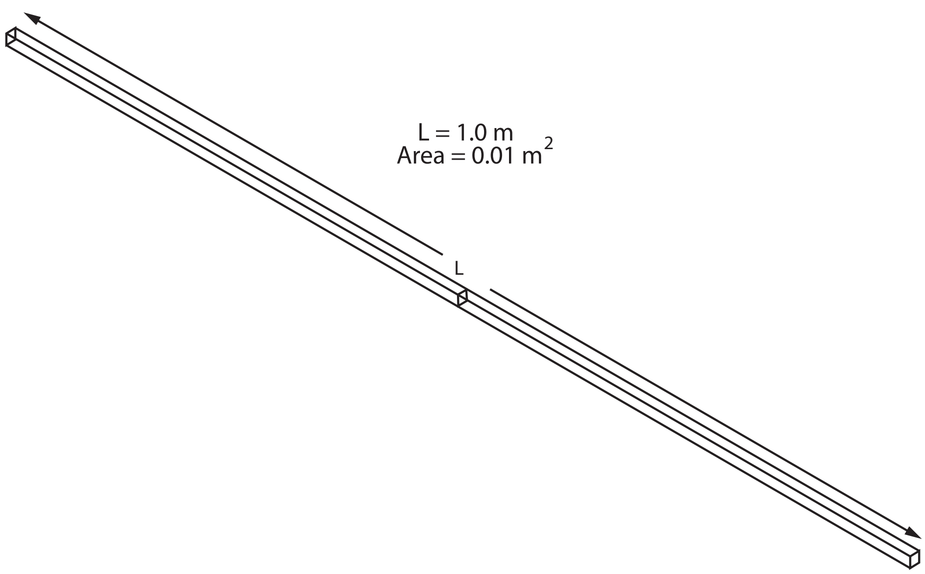


Table 1 Materials, Geometry, and Boundary Conditions

Material Properties	Geometry	Boundary Conditions
Density is specified using the Aungier-Redlich-Kwong real gas model	Length of the tube = 1 m Area of cross section = 0.01 m^2	Cell zone conditions are specified with high pressure and low pressure properties of hydrogen

Analysis Assumptions and Modeling Notes

The flow is compressible and unsteady by nature. Real gas effects are significant in the pressure range considered here.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Static Temperature Along Centerline of the Tube

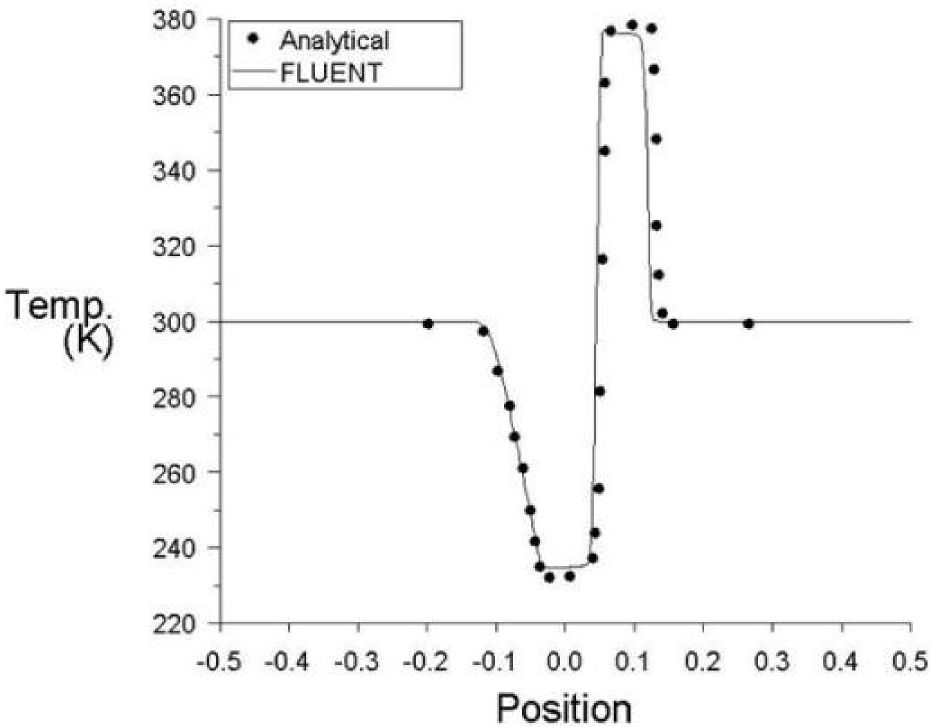
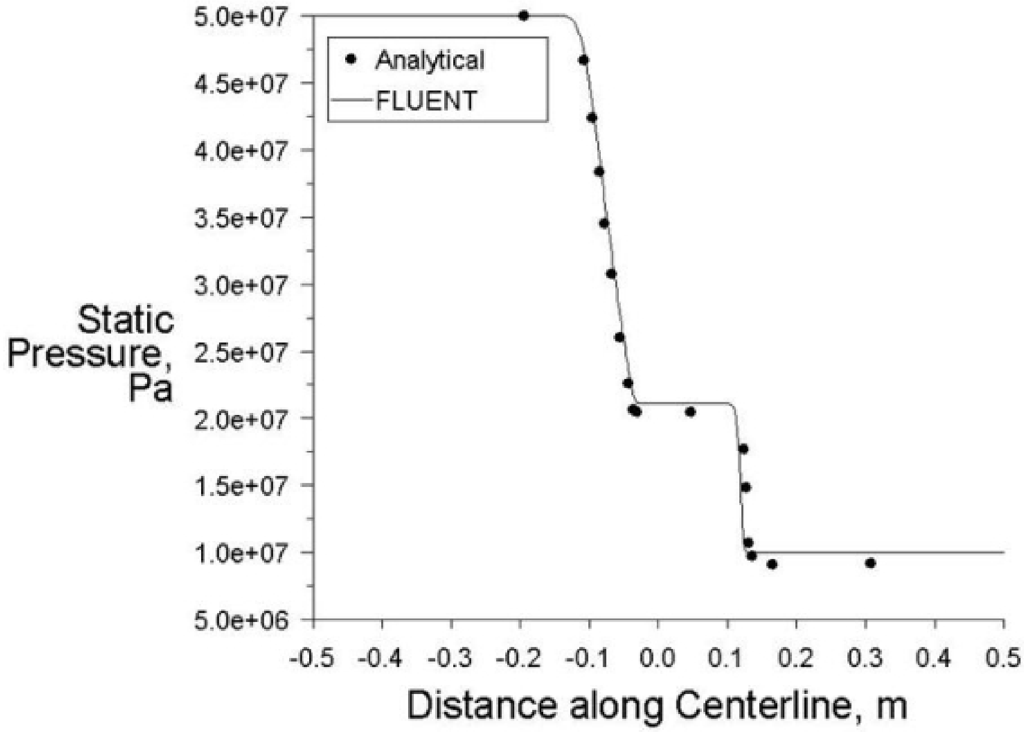


Figure 3 Comparison of Static Pressure Along Centerline of the Tube



Results Comparison for ANSYS CFX

Figure 4 Comparison of Static Temperature Along Centerline of the Tube

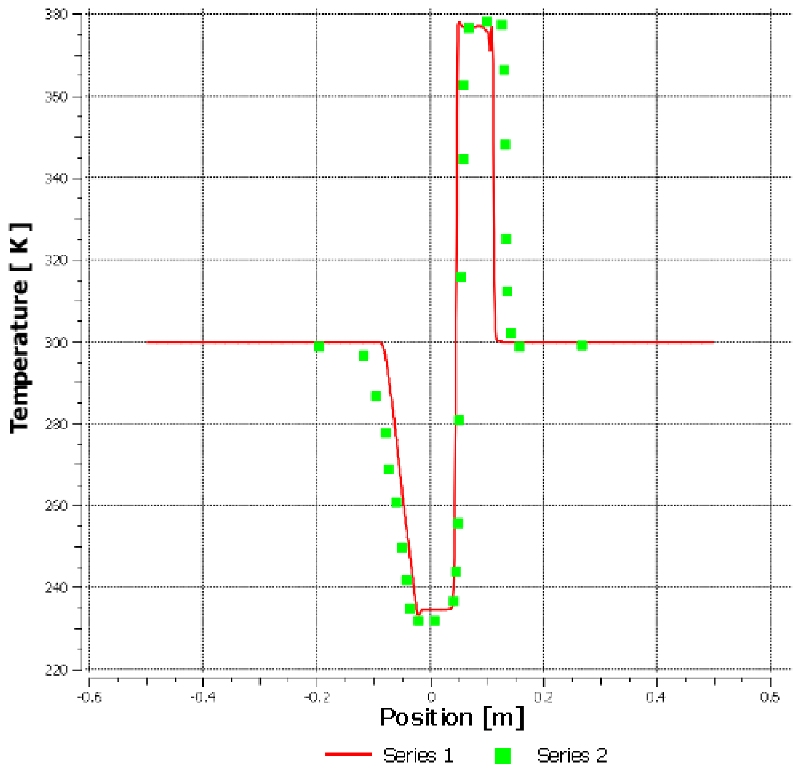
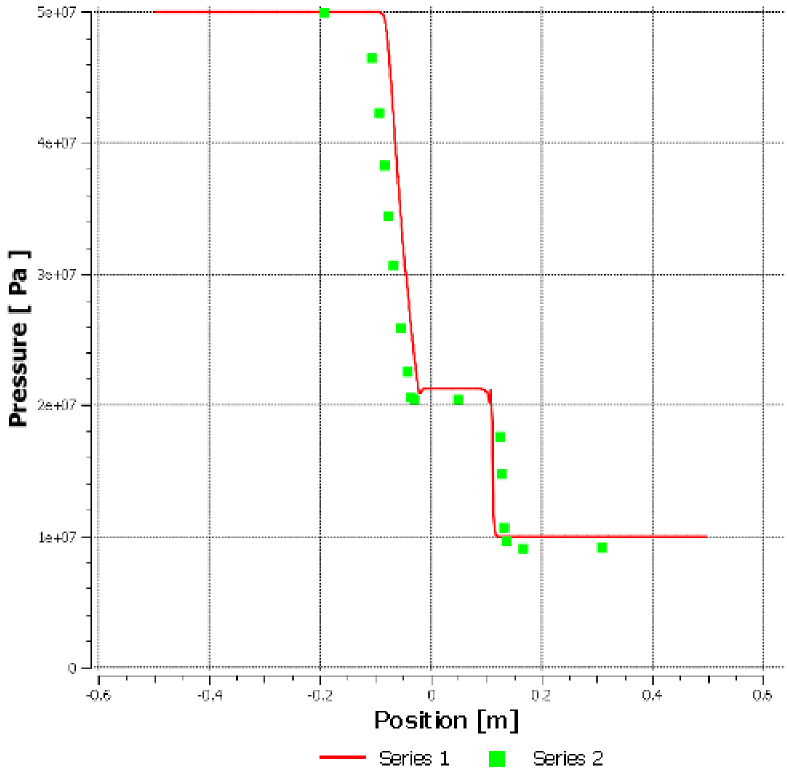


Figure 5 Comparison of Static Pressure Along Centerline of the Tube



VMFL027: Turbulent Flow over a Backward-Facing Step

Overview

Reference	D.M. Driver and H.L. Seegmiller. "*Features of a Reattaching Turbulent Shear Layer in Divergent Channel Flow*". *AIAA Journal*. Vol. 23, No. 2. 163-171. Feb. 1985.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	2-D turbulent flow with separation and reattachment, realizable k-ε model
Input File	**`drivseeg-rke-neqwf.cas`** for ANSYS FLUENT **`VMFL027B_step.def`** for ANSYS CFX

Test Case

Turbulent flow over a backward facing step is modeled. The flow separates at the step and reattaches on the wall downstream, enclosing a region of recirculation. The inlet is at 4 H upstream and the outlet at 30 H downstream from the location of the step, where H is the step height. Reynolds number based on the step-height is about 28,000.

Figure 1 Flow Domain

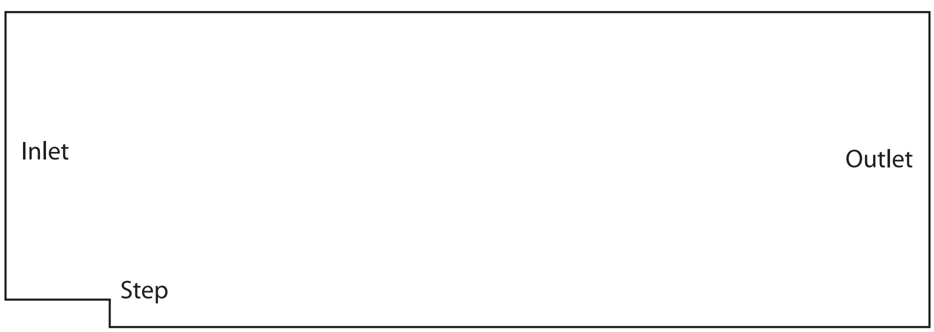


Table 1 Materials, Geometry, and Boundary Conditions

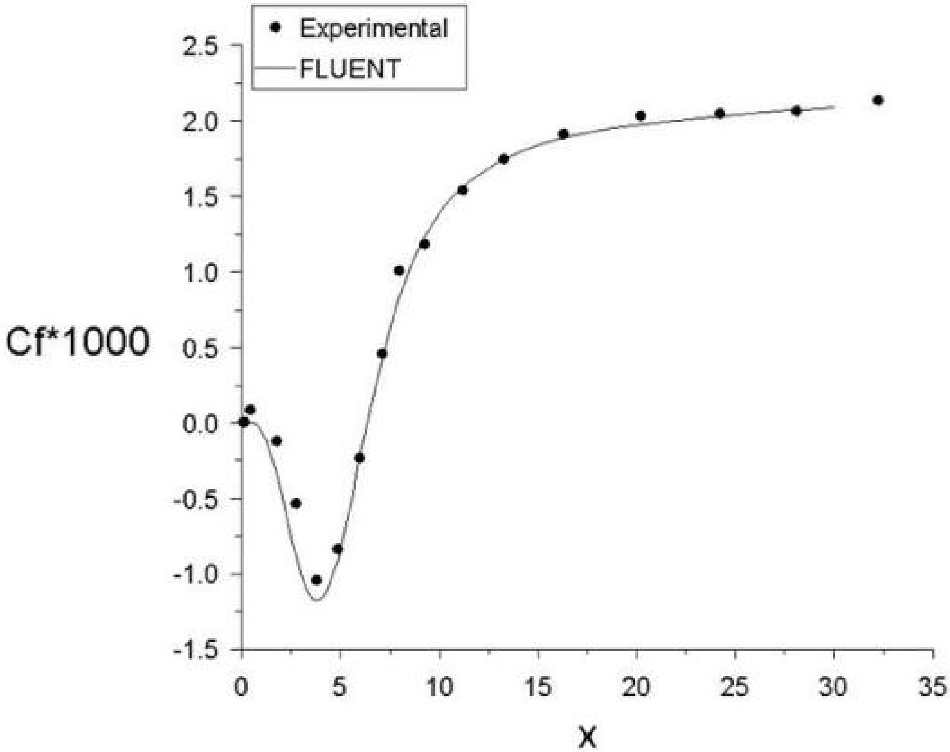
Material Properties	Geometry	Boundary Conditions
Density : 1 kg/m^3 Viscosity: 0.0001 kg/m-s	Step height = 1m Total length of the channel = 34 m Height of the channel = 9 m	Inlet velocity specified as fully developed turbulent velocity profile

Analysis Assumptions and Modeling Notes

The flow is steady. Realizable k-ε model was used to model turbulence.

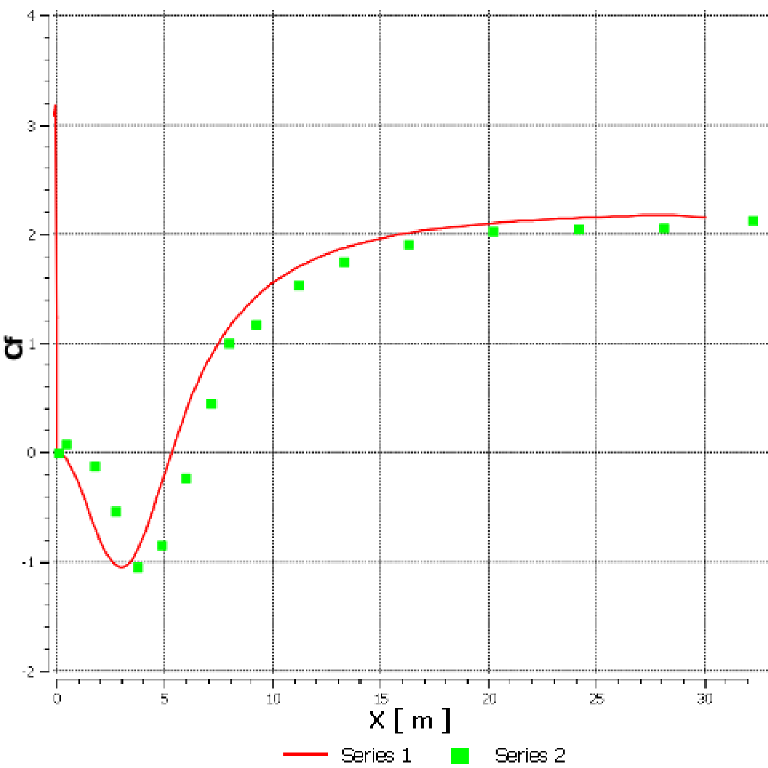
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Skin Friction Coefficient Along the Wall



Results Comparison for ANSYS CFX

Figure 3 Comparison of Skin Friction Coefficient Along the Wall



VMFL028: Turbulent Heat Transfer in a Pipe Expansion

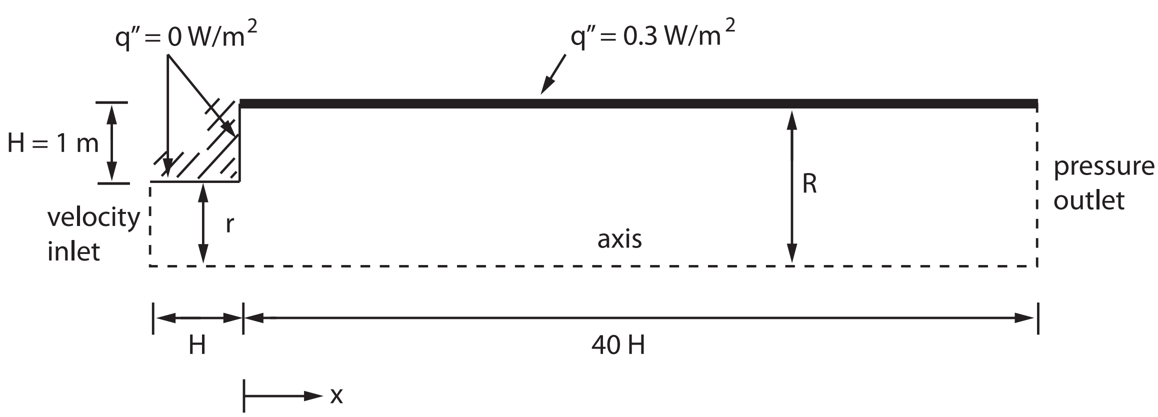
Overview

Reference	Baughn et al..*"Local Heat Transfer Downstream of an Abrupt Expansion in a Circular Channel With Constant Wall Heat Flux". Journal of Heat Transfer*, 106. 789-796. 1984.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Heat transfer, turbulent flow with recirculation and reattachment
Input File	`bghnexp.cas` for ANSYS FLUENT `VMFL028B_pipe_expansion.def` for ANSYS CFX

Test Case

Fully developed turbulent flow through an axisymmetric pipe expansion is modeled. The flow reattaches to the pipe wall downstream of the expansion, enclosing a zone of recirculation. The pipe wall downstream of the expansion is heated at a constant rate. Inlet to the computational domain is placed at 1 step height upstream of the expansion and the outlet at 40 step-heights downstream.

Figure 1 Flow Domain



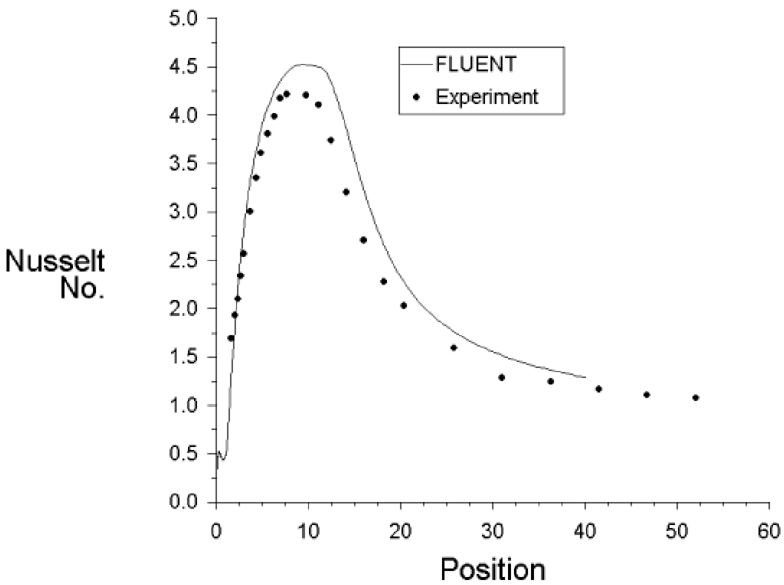
Material Properties	Geometry	Boundary Conditions
Density: 1.225 kg/m^3 Viscosity: 1.68318e-5kg/m-s Specific Heat: 1006.43 J/kg-K Thermal Conductivity: 0.0242 W/m-K	Pipe radius before expansion = 0.667 m Pipe radius after expansion = 1.6667 m	Inlet velocity: Specified by fully developed turbulent velocity profile Inlet temperature = 273 K Heat flux across the wall after expansion = 0.3 W/m^2

Analysis Assumptions and Modeling Notes

Steady flow in axisymmetric domain. The wall upstream of expansion is adiabatic.

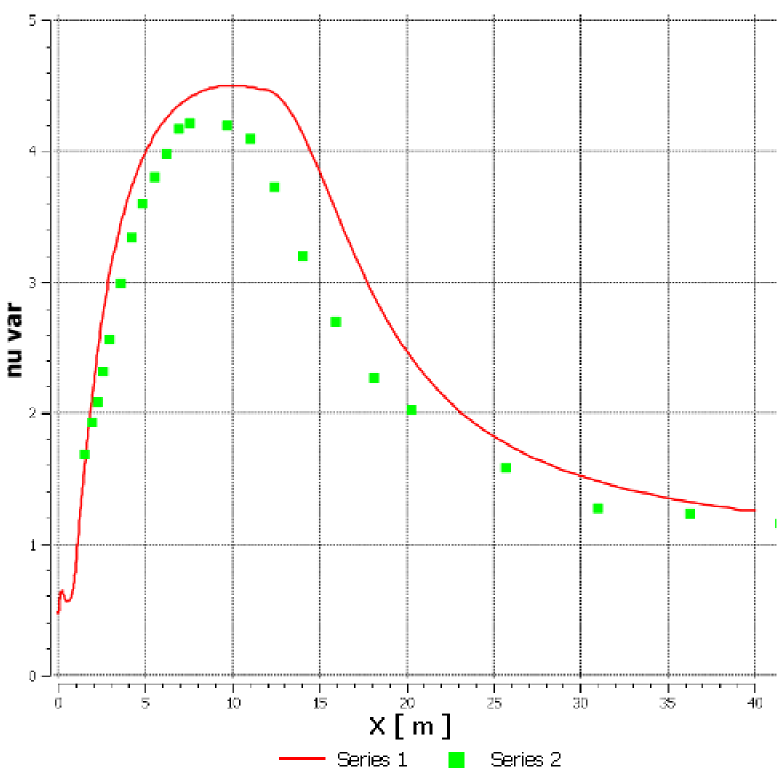
Results Comparison for ANSYS FLUENT

Figure 2 Nusselts Number Variation along the Heat Wall



Results Comparison for ANSYS CFX

Figure 3 Nusselts Number Variation along the Heat Wall



VMFL029: Anisotropic Conduction Heat Transfer

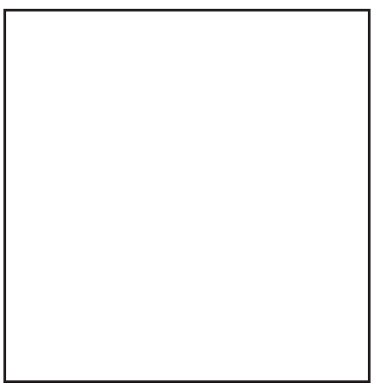
Overview

Reference	
Solver	ANSYS FLUENT (ANSYS CFX simulation is not available for this case)
Physics/Models	Heat conduction, anisotropic conductivity
Input File	`aniso.cas`

Test Case

Heat conduction in a solid with anisotropic thermal conductivity is modeled. A square domain is considered. Two opposite walls are maintained at uniform temperatures. Conductivity of the solid material is specified using matrix components to account for the anisotropy. The simulation results are compared with analytical solution for temperature distribution.

Figure 1 Domain



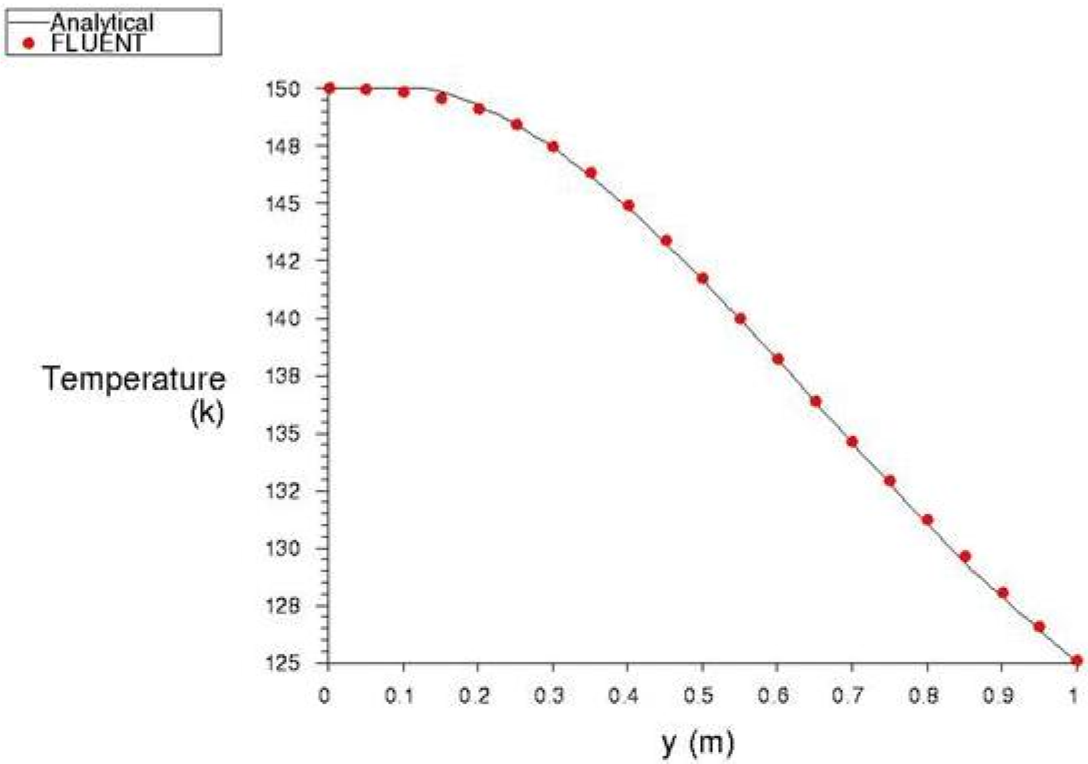
Material Properties	Geometry	Boundary Conditions
Density of solid = 2719 kg/m^3 Specific heat = 871 J/kg-K Thermal conductivity: Anisotropic	Dimensions of the domain: 1m X 1 m	Fixed wall temperatures = 100 K and 200K respectively User-defined profile for temperature distribution on the other two walls

Analysis Assumptions and Modeling Notes

Steady state conduction. Anisotropic conductivity modeled by specified matrix components for the solid conductivity.

Results Comparison

Figure 2 Comparison of Temperature Distribution at X = 0.5 m



VMFL030: Turbulent Flow in a 90° Pipe-Bend

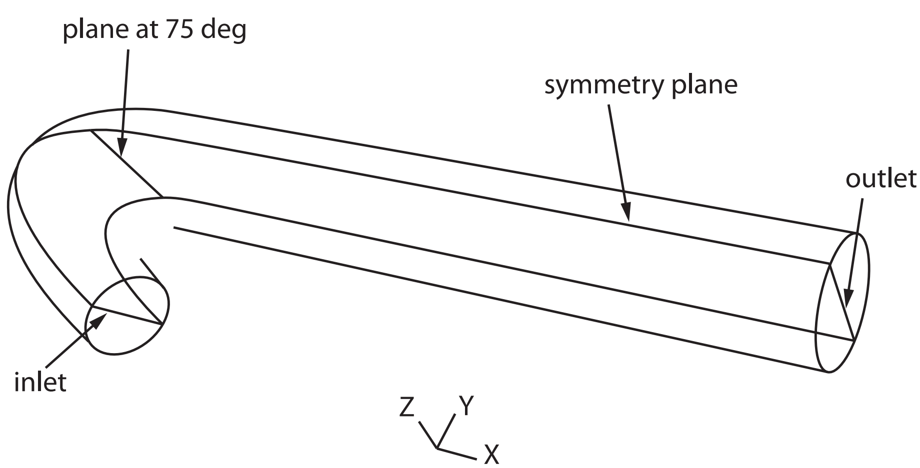
Overview

Reference	M. M. Enayet, M. M. Gibson, A. M. K. P. Taylor, and M. Yianneskis. "*Laser-Doppler Measurements of Laminar and Turbulent Flow in a Pipe Bend*". *Znt. J. Heat & Fluid Flow*. Vol. 3. 213-219. 1982.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	3–D Turbulent flow with separation, RNG k-ε model with non-equilibrium wall functions
Input File	`pipebnd-rng-noneq.cas` for ANSYS FLUENT `VMFL030B_vv030.def` for ANSYS CFX

Test Case

Turbulent flow through a 90° circular pipe bend is modeled. The flow separates and reattaches around the bend. Due to symmetry of the flow field only half of the domain is modeled. Velocity profile at an angle of 75° (as measured from the inlet) is used to validate the simulation.

Figure 1 Flow Domain



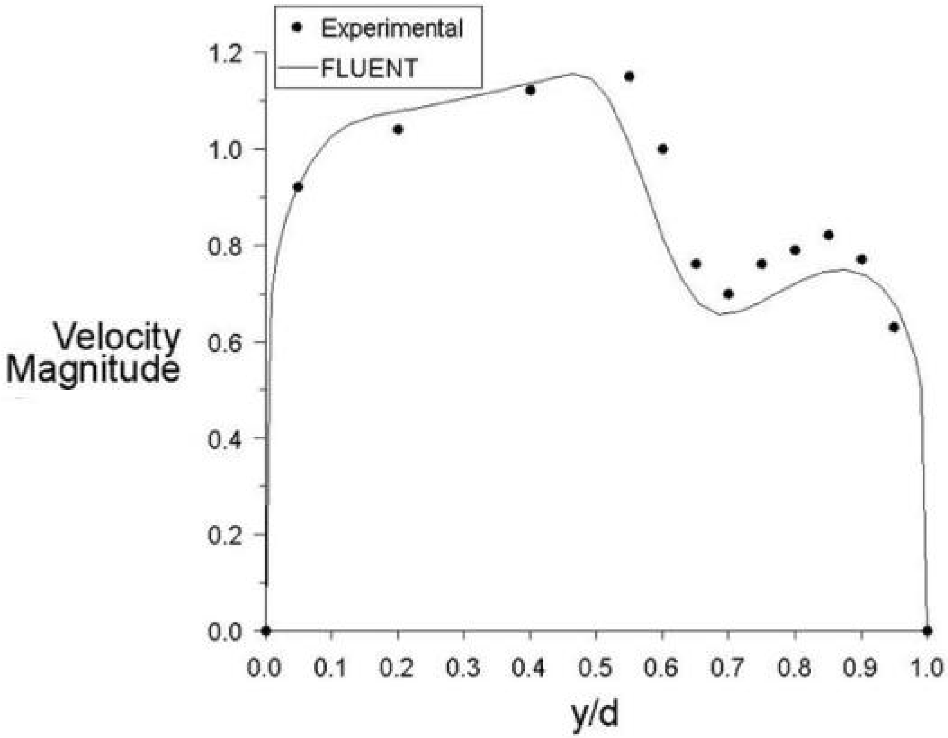
Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 2.3256 x 10^{-05} kg/m-s	Radius of the pipe = 0.5 m	Inlet velocity: Fully developed turbulent profile for z-velocity. Non components in other directions

Analysis Assumptions and Modeling Notes

The flow is steady. RNG k- ε is used to model turbulence along with non-equilibrium wall functions.

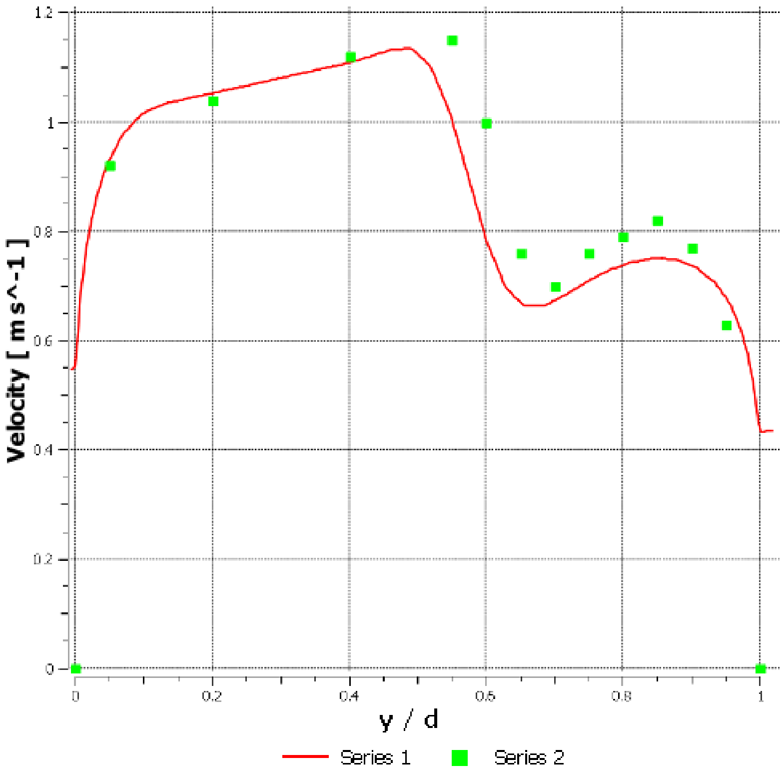
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Velocity Magnitude (m/s) at 75° Along the Bend



Results Comparison for ANSYS CFX

Figure 3 Comparison of Velocity Magnitude (m/s) at 75° Along the Bend



VMFL031: Turbulent Flow Behind an Open-Slit V Gutter

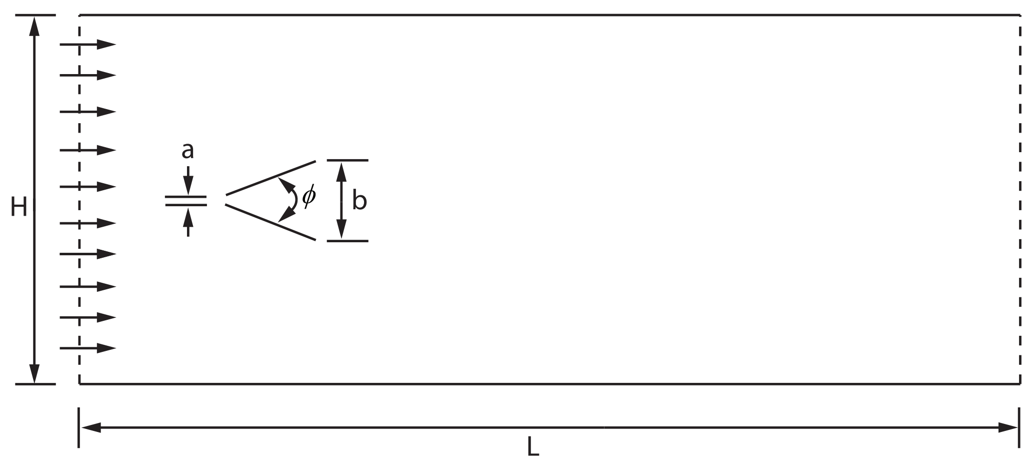
Overview

Reference	J.-T. Yang and G.-L. Tsai. "*Near-wake flow of a v-gutter with slit bleed*". *Journal of Fluid Engineering*. Vol. 115. 13-20. March, 1993.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent flow
Input File	**`spltvee.cas`** for ANSYS FLUENT **`VMFL031B_veesplit.def`** for ANSYS CFX

Test Case

The near-wake flow structure behind an open-slit V gutter at airflow speed of 20 m/s is modeled. The interaction between the flow penetrating through the open slit and the shear layer results in an asymmetric wake flow structure. The size of the entire recirculation zone shifts toward one of the two wings due to the Coand effect.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 1.8333 X 10^{-05} kg/m-s	L = 40 cm H = 10 cm a = 2 mm b = 22 mm ø= 45°	u_{inlet} = 20 m/s k_{inlet} = 0.04335 m^2/s^2 ε_{inlet} = 0.2119 m^2/s^3

Analysis Assumptions and Modeling Notes

Steady, turbulent, incompressible flow. The standard k-ε model is used for turbulence.

Results Comparison for ANSYS FLUENT

The x-velocity at x = 22 mm downstream of the split-V-gutter, is compared with experimental data.

Figure 2 X-Velocity at x = 22 mm Downstream of the V-Gutter

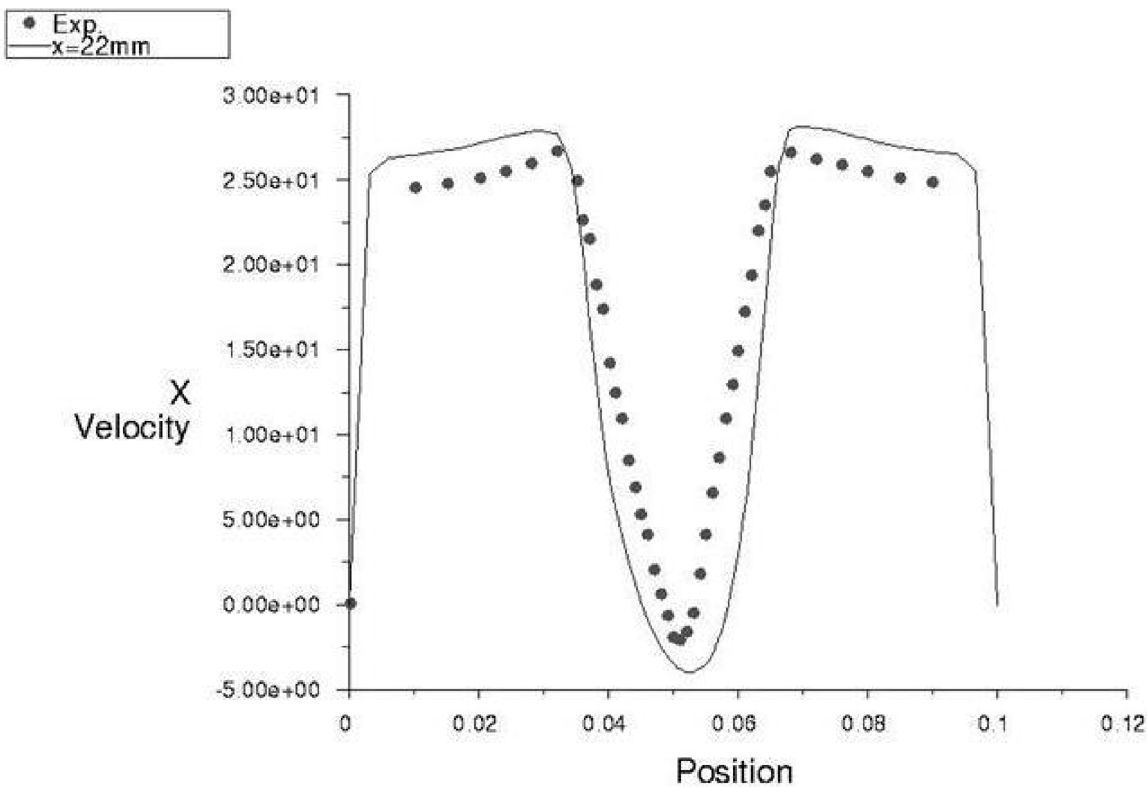
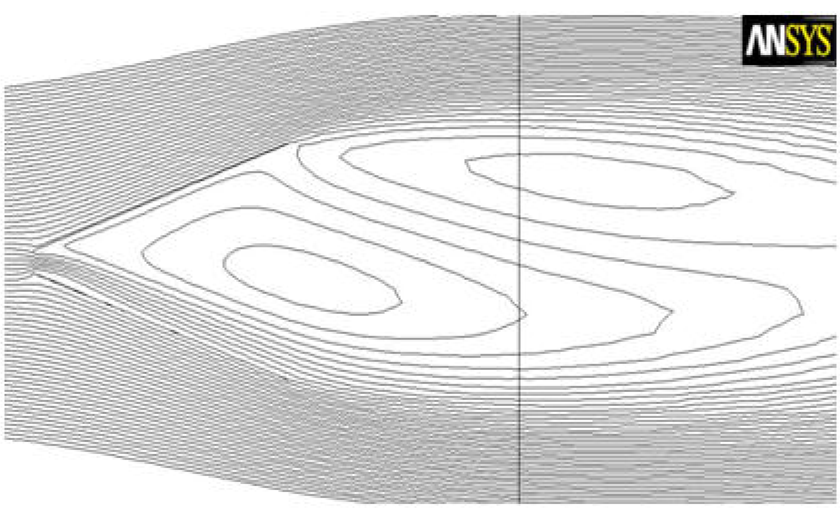
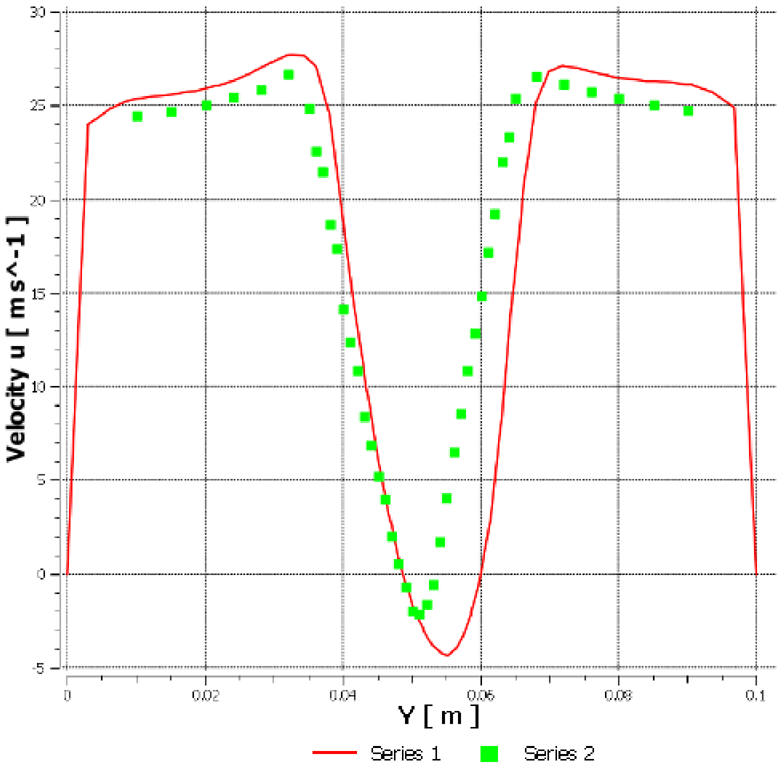


Figure 3 The Coand Effect



Results Comparison for ANSYS CFX

Figure 4 X-Velocity at x = 22 mm Downstream of the V-Gutter



VMFL032: Turbulent Flow with Separation Along an Axisymmetric Afterbody

Overview

Reference	T.T. Huang and N.C. Groves. "*Propeller/stern boundary layer interaction on axisymmetric bodies: Theory and experiment*". *David W. Taylor Naval Ship Research and Development Center Rep*. 76-0113. 1976.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent flow
Input File	`axiaft.cas` for ANSYS FLUENT `VMFL032B_afterbody.def` for ANSYS CFX

Test Case

Flow past an axisymmetric afterbody, representing the hull of ship. The flow separates on the rear face of the body.

Figure 1 Flow Domain

Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 1 X 10^{-6} kg/m-s	Length of the afterbody = 1.0 m Maximum radius of the afterbody = 0.04556 m	Fully developed turbulent velocity profile on the inlet normal to axis Axial velocity = 5.9 m/s on the inlet parallel to axis

Analysis Assumptions and Modeling Notes

The far-field boundary of the domain is set parallel to the axis and is modeled as velocity inlet. Fully developed profile is specified at the transverse velocity inlet.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Pressure Coefficient Along the Afterbody Wall

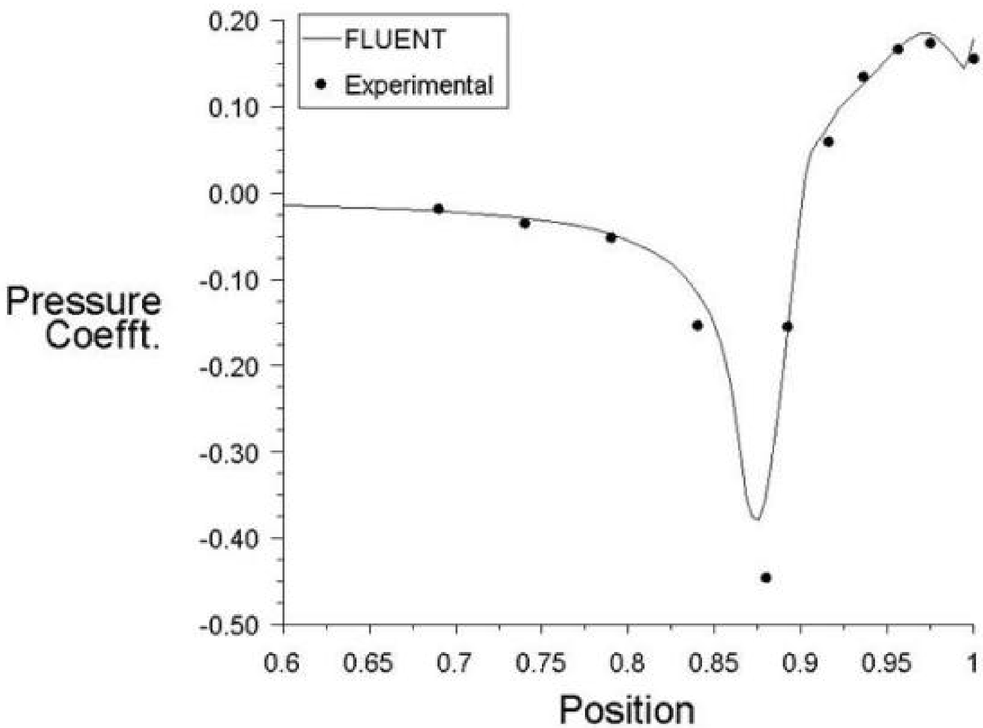
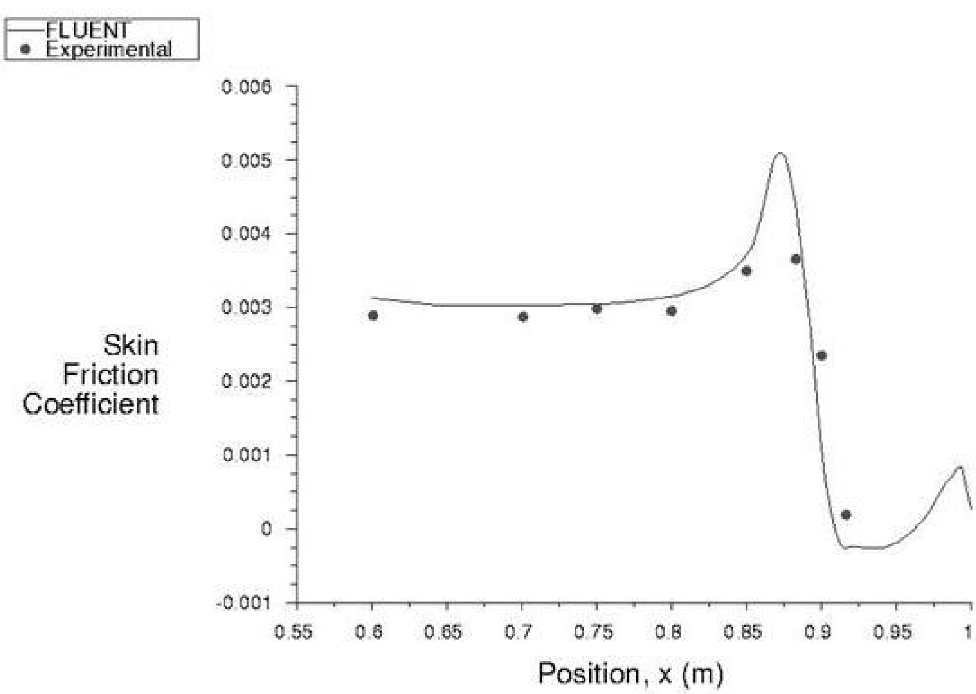


Figure 3 Comparison of Skin Friction Coefficient Along the Afterbody Wall



Results Comparison for ANSYS CFX

Figure 4 Comparison of Pressure Coefficient Along the Afterbody Wall

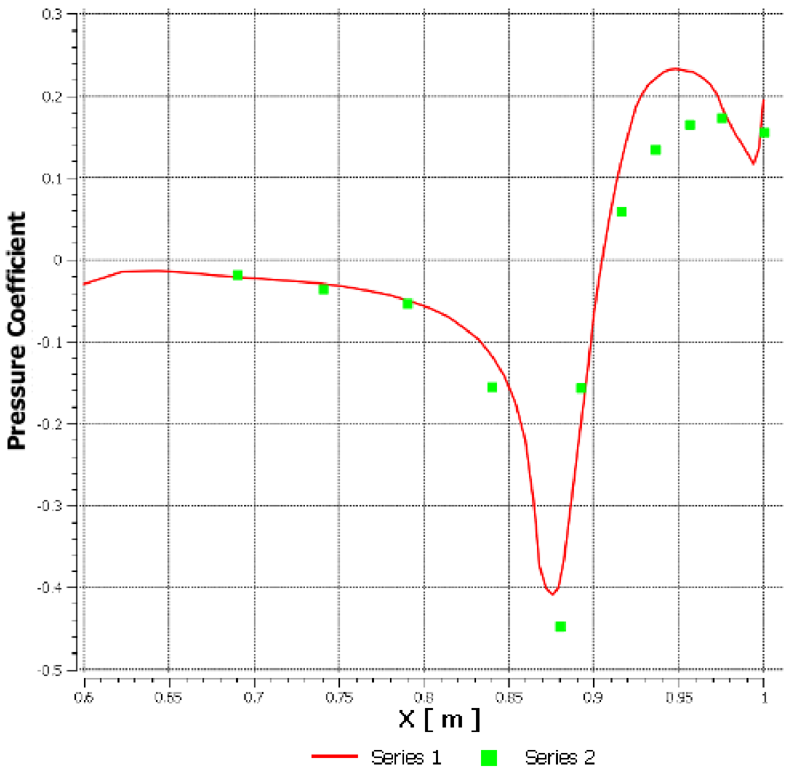
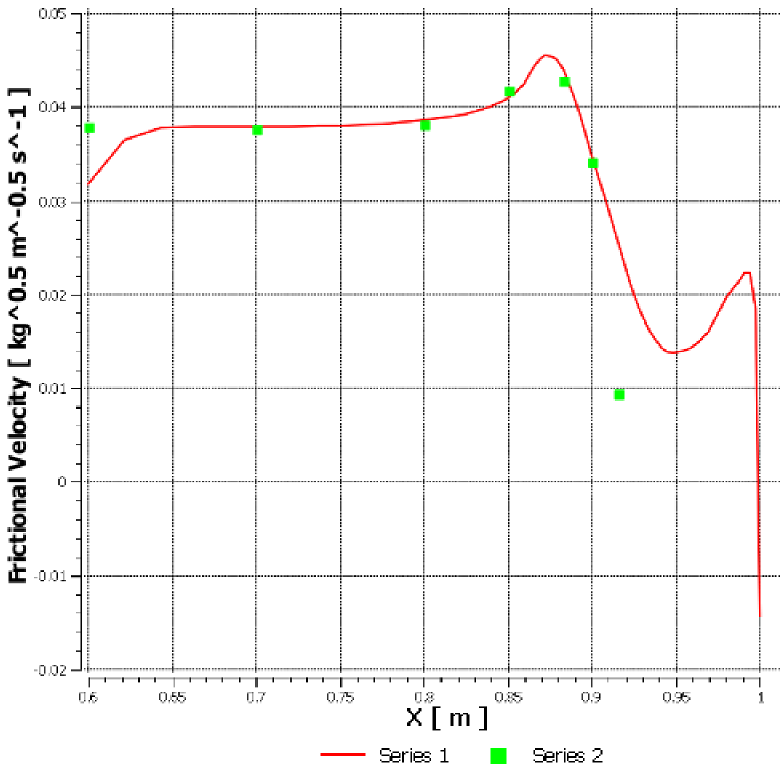


Figure 5 Comparison of Skin Friction Coefficient Along the Afterbody Wall



VMFL033: Viscous Heating in an Annulus

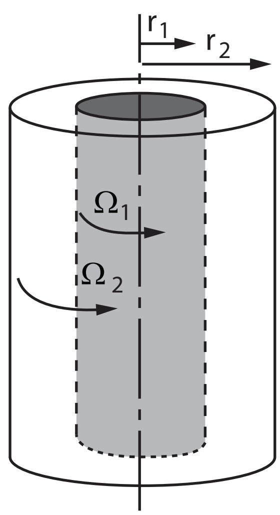
Overview

Reference	R. B. Bird, W. E. Stewart, and E. N. Lightfoot. "*Transport phenomena*". John Wiley and Sons. New York. 1960.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Viscous flow and heating driven by a moving wall
Input File	`plate_polar.cas` for ANSYS FLUENT `VMFL033B_CFX.def` for ANSYS CFX

Test Case

In this problem, we model the viscous heating and mass flow in a 2-D annulus induced by the rotation of one of the two walls (*Figure 1* (p. 125)). This problem can be solved analytically.

Figure 1 Geometry



Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Specific heat: 1 J/kg-K Thermal conductivity: 1 W/m-K Viscosity: 300 kg/m-s	r_1= 1 m r_2 = 2 m	Ω_1 = 0.0 rad/s Ω_2 = 0.5 rad/s T_1 = 273 K T_2 = 274 K

Analysis Assumptions and Modeling Notes

The flow is laminar and steady. Pressure based solver is used. A 2–D mesh with quadrilateral cells is used to discretize the domain.

Results Comparison for ANSYS FLUENT

Normalized velocity and temperature profiles are compared with the analytical solution provided by Bird et al (1960).

Figure 2 Comparison of Velocity Profile

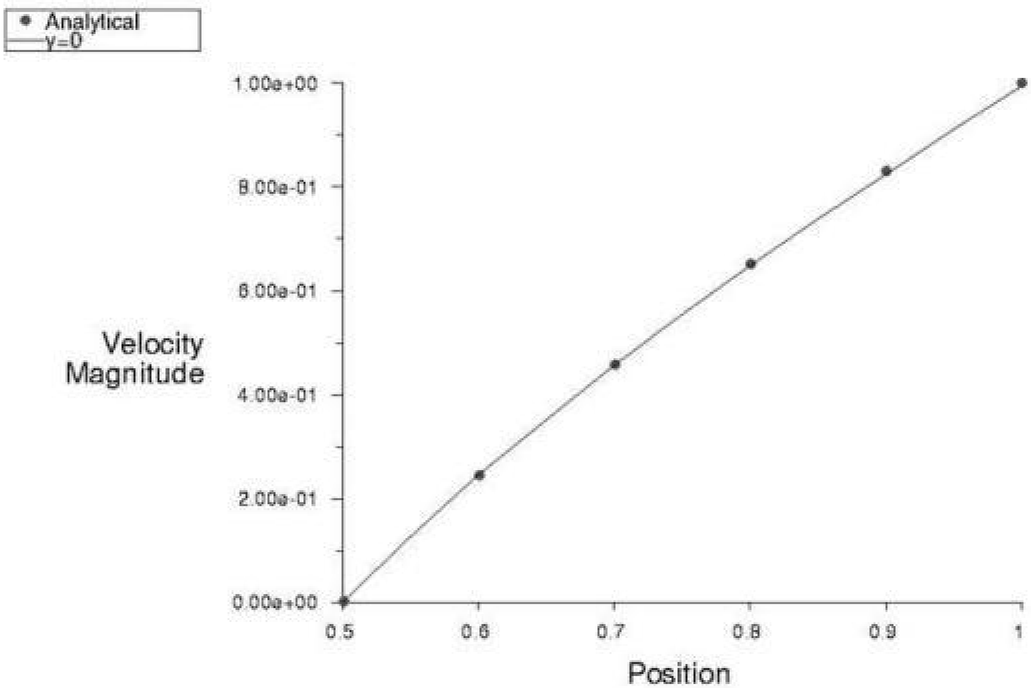
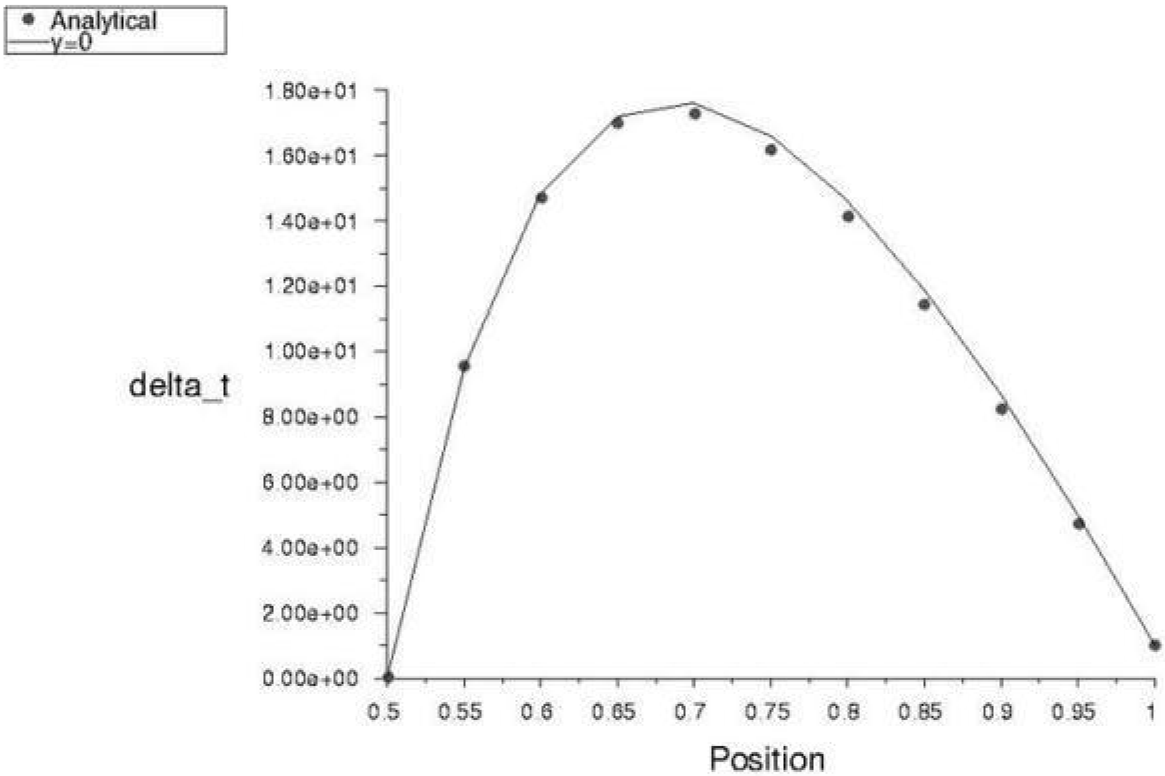


Figure 3 Comparison of Temperature Profile



Results Comparison for ANSYS CFX

Figure 4 Comparison of Velocity Profile

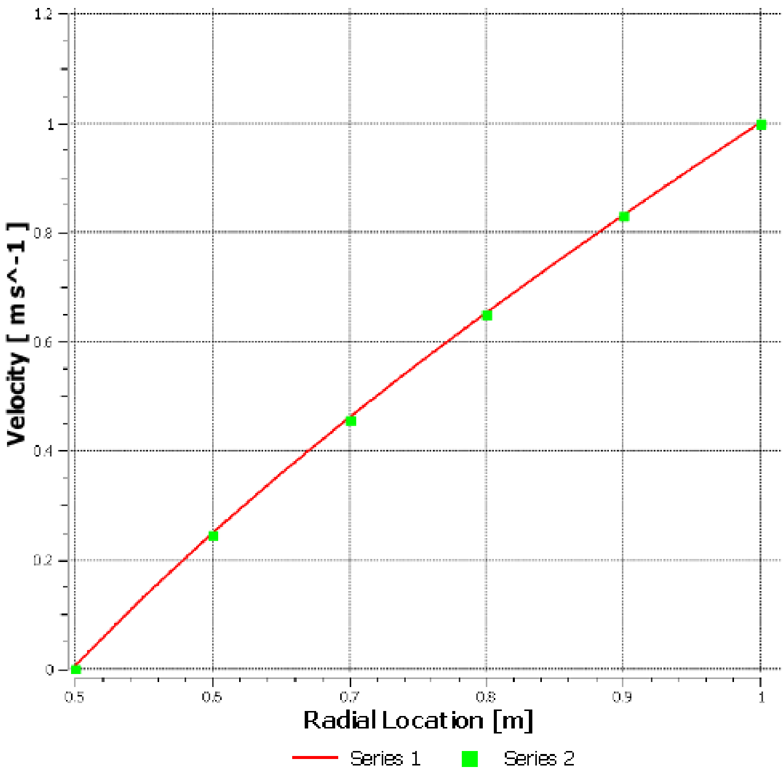
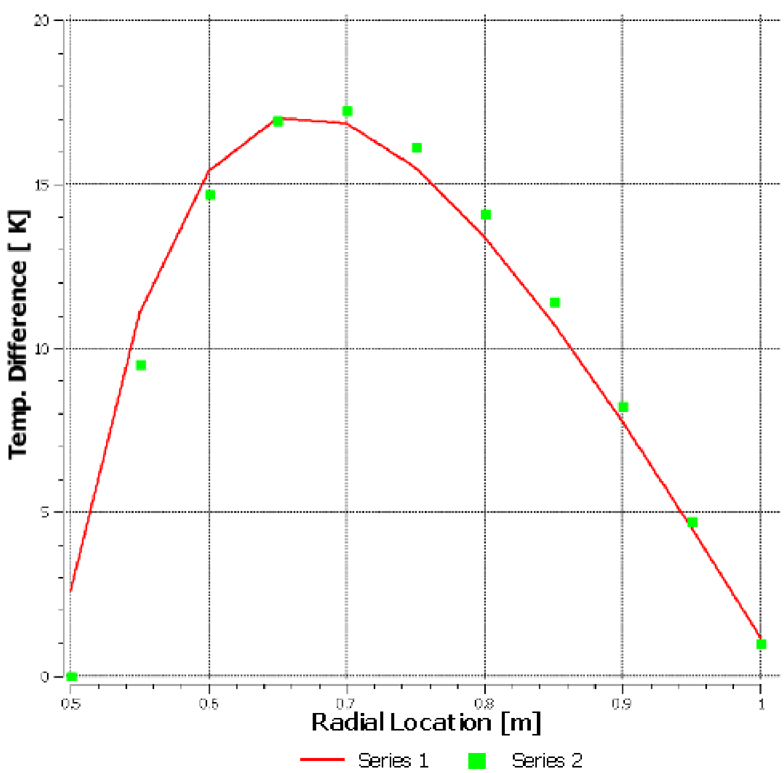


Figure 5 Comparison of Temperature Profile



VMFL034: Particle Aggregation inside a Turbulent Stirred Tank

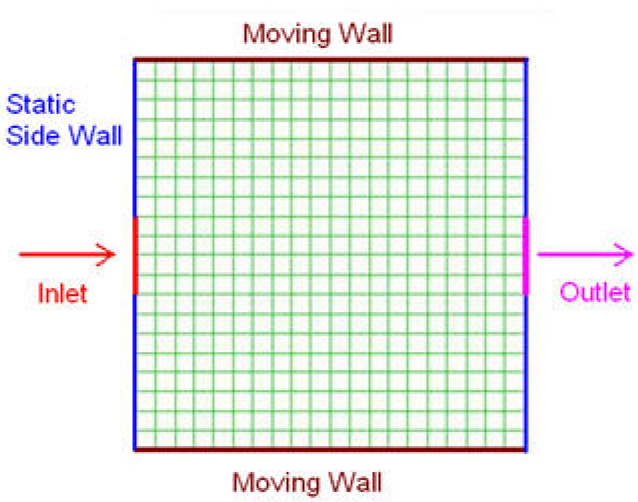
Overview

Reference	B. Wan, T.A. Ring, K. Dhanasekharan, and J. Sanyal. "*Comparison of Analytical Solutions for CMSMPR Crystallizer with QMOM Population Balance Modeling in ANSYS FLUENT*". *China Particuology*. Vol. 3(4). 213-218. 2005.
Solver	ANSYS FLUENT (ANSYS CFX simulation is not available for this case)
Physics/Models	Multi-phase, Population balance model, turbulent flow
Input File	`agglomeration.cas`

Test Case

A 2-D approximation of a stirred tank is simulated in order to verify the population balance model that operates in conjunction with its multiphase calculations to predict the particle size distribution within the flow field. The flow rate at the inlet is equal to that at the outlet, allowing the mean residence time to be calculated from the inlet flow rate (velocity x inlet area) and the "volume" (box area x unit depth) of the box. To simulate the agitation in the tank the top and bottom walls are assumed to move in the direction of the outlet. The flow is turbulent, steady, and incompressible. Multi-phase, with QMOM population balance model is used for particle aggregation. The standard k-ε model is used for turbulence.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density: 998.2 kg/m^3 Viscosity: 0.00103 kg/m-s	Square box side = 0.1 m Inlet/Outlet openings = 0.02 m	Top wall velocity: 101 m/s Bottom wall velocity: 100 m/s Inlet velocity = 0.005 m/s Outlet gauge pressure = 0 Pa

Analysis Assumptions and Modeling Notes

The results of the ANSYS FLUENT simulation are compared to steady state analytical solution for the population balance in a stirred tank where aggregation takes place.

Results Comparison

In this table, moment of PBE for ANSYS FLUENT turbulent simulations is compared with analytical solution for aggregation alone at the outlet of the tank.

Table 1 Comparison of Moment of PBE

Moment	Target	ANSYS FLUENT	Ratio
m_0	0.132	0.132	1.000
m_1	0.225	0.226	1.004
m_2	0.547	0.548	1.002
m_3	1.910	1.910	1.000
m_4	9.073	9.133	1.007
m_5	53.797	53.816	1.000

VMFL035: 3-Dimensional Single-Stage Axial Compressor

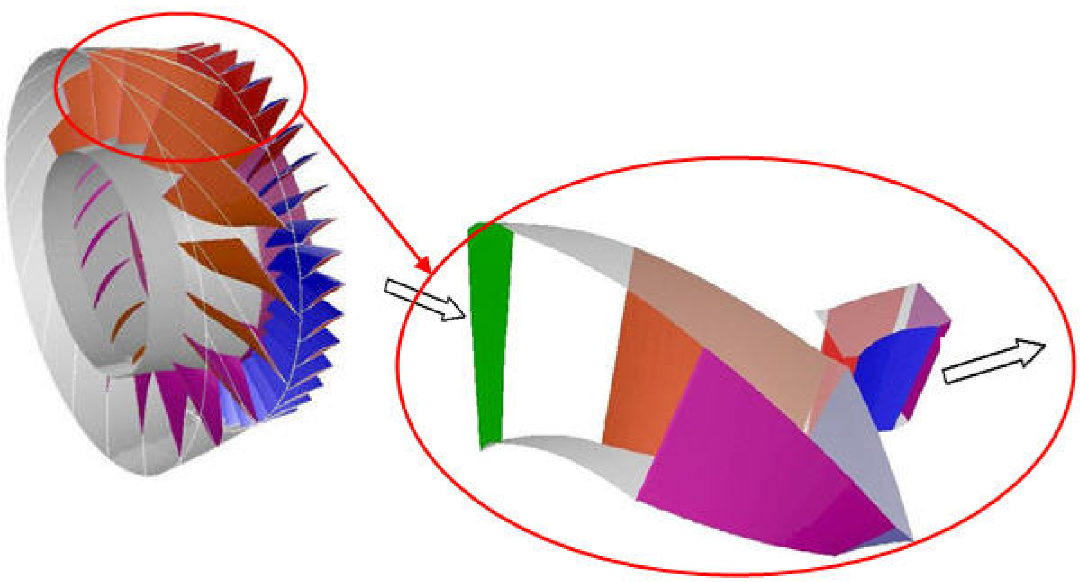
Overview

Reference	Density-based solver (ANSYS FLUENT)
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Compressible (transonic), turbulent flow, moving reference frame
Input File	`axial-compressor.cas`

Test Case

A 3-D model of a single-stage axial compressor is simulated. The flow through the rotor blades is computed in a rotating reference frame, while the flow in the stator blades in a stationary frame. The purpose of this case is to validate the performance of the pressure-based coupled solver for a compressible turbomachinery problem with a mixing plane. The flow is compressible, turbulent and steady.

Figure 1 Flow Domain



Material Properties (for Air)	Geometry	Boundary Conditions
Density = Ideal – Gas Molecular weight = 28.966 Specific heat = 1006.43 J/kg-K Viscosity, Thermal conductivity: Kinetic theory	Geometry is as shown in in *Figure 1* (p. 129) Number of rotor blades = 16 Number of stator blades = 40	Rotational speed = 37,500 rpm **For Inlet:** • P_{total} = 1 atm • T_{total} = 288 K

Analysis Assumptions and Modeling Notes

Steady, turbulent, compressible flow. Ideal-gas law is used for density calculations and kinetic theory for fluid viscosity and thermal conductivity. The standard k-ε model is used for turbulence. Pressure-based coupled solver with a mixing plane at the rotor-outlet/stator-inlet interface.

Results Comparison for ANSYS FLUENT

The results of the pressure-based ANSYS FLUENT simulation are compared to the steady state solution from the density-based solver.

Table 1 Comparison of Pressure and Mass Flow Rate

	Target	ANSYS FLUENT	Ratio
Pressure at Stator-Outlet (atm)	1.4725	1.4822	1.007
Mass-Flow Rate at Stator-Outlet (kg/s)	0.1049	0.1076	1.026

Results Comparison for ANSYS CFX

Table 2 Comparison of Pressure and Mass Flow Rate

	Target	ANSYS CFX	Ratio
Pressure at Stator-Outlet (atm)	1.4725	1.4754	1.0020
Mass-Flow Rate at Stator-Outlet (kg/s)	0.1049	0.1078	1.0276

VMFL036: Turbulent Round Jet

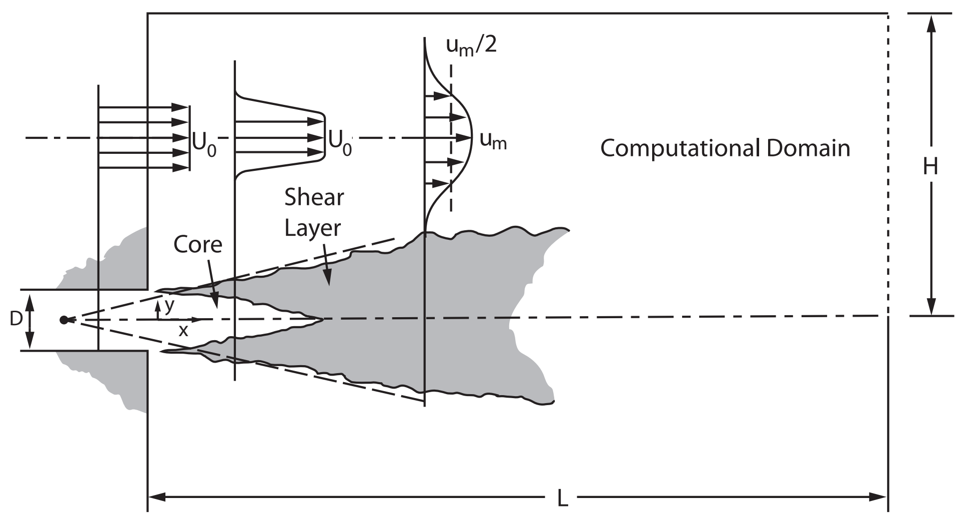
Overview

Reference	D. C. Wilcox. "*Turbulence Modeling for CFD*". DCW Industries, Inc.. 1993.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent flow
Target File	`axjet.cas`

Test Case

A turbulent round jet is defined by a velocity inlet adjacent to a symmetry boundary, and exhausts into a rectangular domain or plenum. The domain is chosen large enough and the boundary does not interfere with the jet. The flow is turbulent and steady. The purpose of this case is to validate the performance of the Reynolds Stress Model for turbulence.

Figure 1 Round Jet Geometry



Material Properties	Geometry	Boundary Conditions
Density = 1 kg/m^3 Viscosity = 1e-05 kg/m-s	D = 1 m L = 50 m H = 20 m	Inlet: U_0 = 1 m/s Turbulent intensity = 1% Turbulent length scale = 1 m Outlet: p_{out} = 1 atm

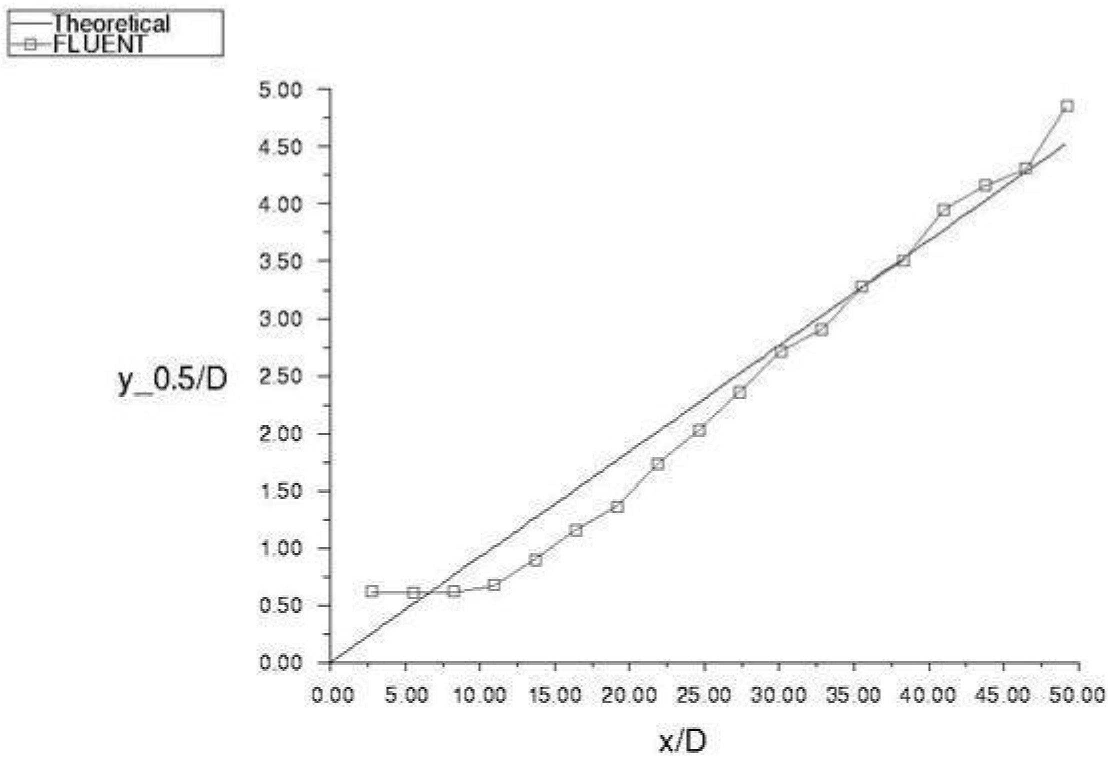
Analysis Assumptions and Modeling Notes

Steady, turbulent, incompressible flow. The Reynolds Stress Model (RSM) is used for turbulence.

Results Comparison for ANSYS FLUENT

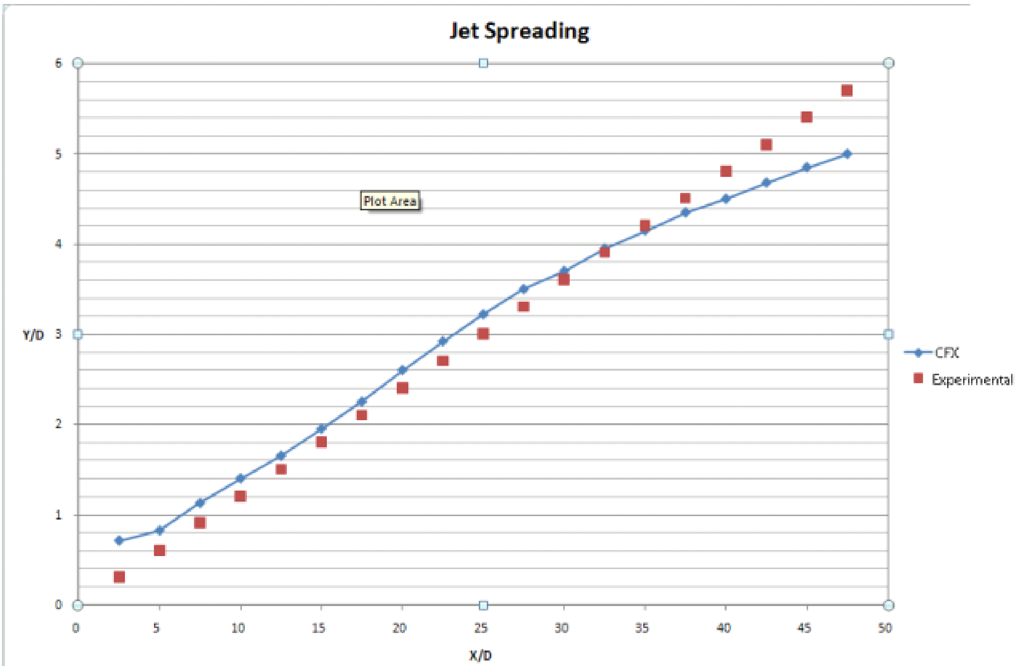
The jet's spreading rate is compared to the Wilcox (1998) data for round jets. The scattering of the data is due to the coarse triangular grid that has been used in this study. A finer grid would have produced a much smoother computational curve.

Figure 2 Comparison of Results for ANSYS FLUENT



Results Comparison for ANSYS CFX

Figure 3 Comparison of Results for ANSYS CFX



VMFL037: Turbulent Flow over a Forward Facing Step

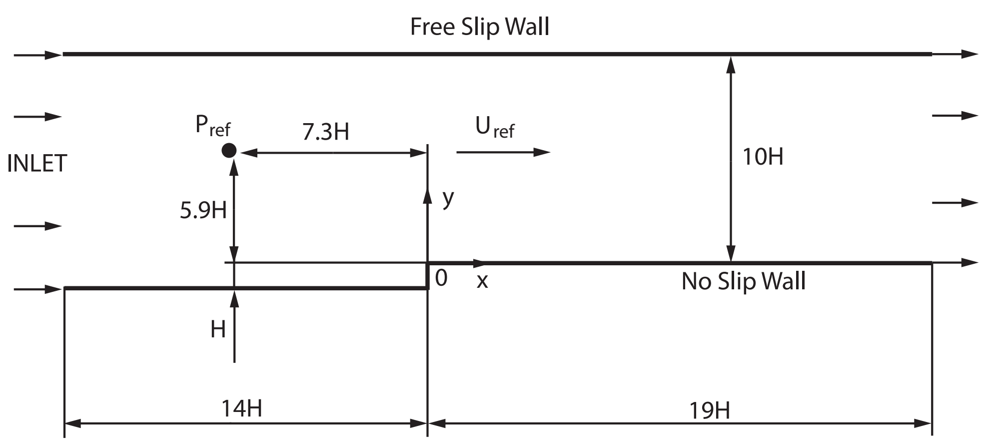
Overview

Reference	S. Baker. "*Regions of Recirculating Flow Associated with Two-Dimensional Steps*". Ph.D. thesis. Department of Civil Engineering, University of Surrey. UK. 1977.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	SST model, turbulent flow with separation and reattachment
Input File	`VMFL037_ffstep.cas` for ANSYS FLUENT `ffstep.def` for ANSYS CFX

Test Case

Turbulent flow over a forward facing step is modeled. The flow undergoes separation and reattachment.

Figure 1 Flow Domain



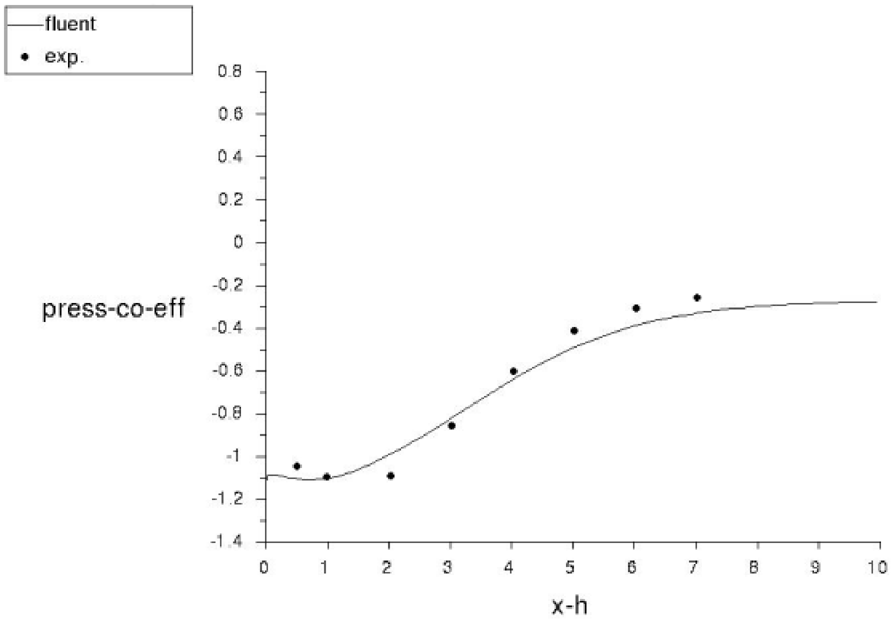
Material Properties	Geometry	Boundary Conditions
Density = 1.02 kg/m^3 Viscosity = 1.5 X 10^{-5} kg/m-s	Step height H = 0.0758 m	Inlet Velocity = 9.7 m/s Outer boundary (in transverse direction) is modeled as slip wall

Analysis Assumptions and Modeling Notes

The flow is steady. Pressure coefficient, Cp on the wall is calculated with reference to the pressure at point upstream of the step at coordinates as indicated in *Figure 1* (p. 137).

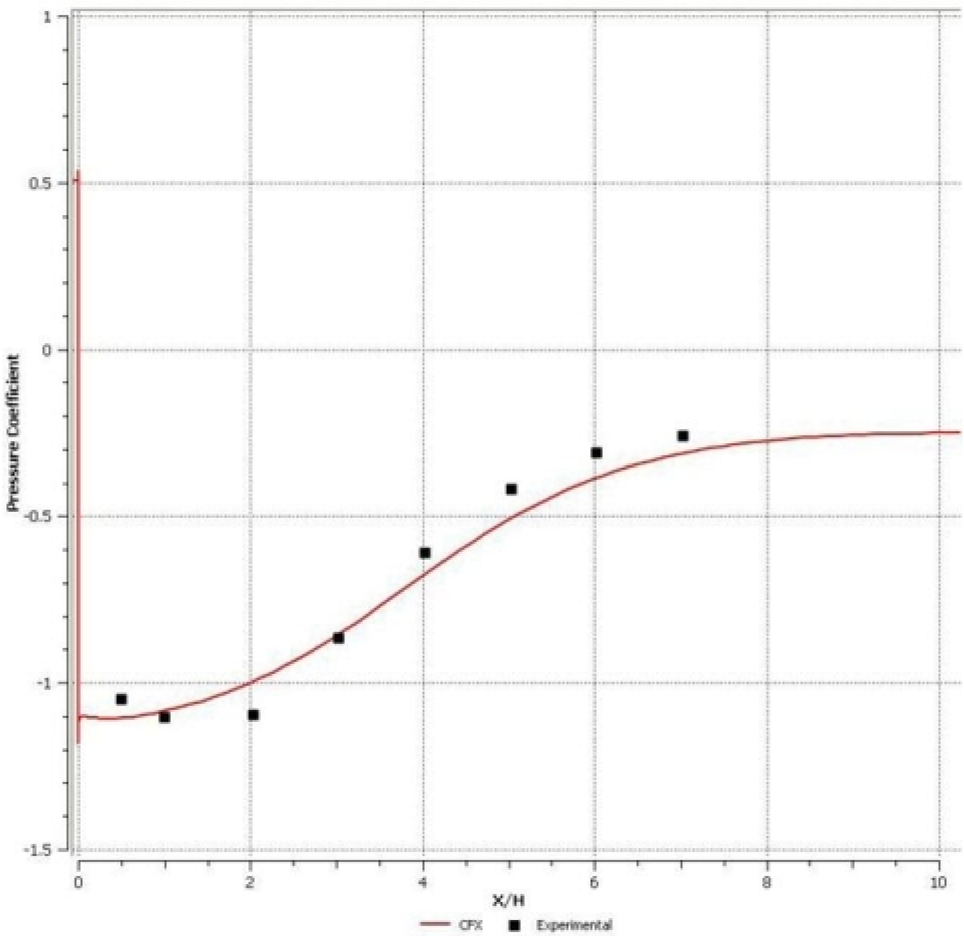
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Pressure Coefficient Along the Wall



Results Comparison for ANSYS CFX

Figure 3 Comparison of Pressure Coefficient Along the Wall



VMFL038: Falling Film over an Inclined Plane

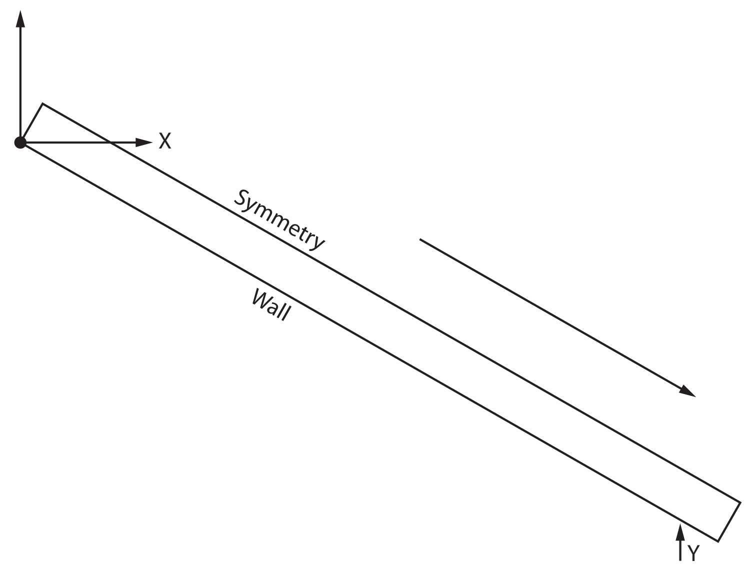
Overview

Reference	RB Bird. "*Transport Phenomena*". WE Stewart and EN Lightfoot. Pg. 45. 2005
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Laminar Flow, Coupled solver
Input File	**`VMFL038_FLUENT.cas`** for ANSYS FLUENT **`VMFL038_CFX.def`** for ANSYS CFX

Test Case

Laminar flow of a fluid over an inclined plane, driven by the pressure difference due to gravity head is modeled. The flow channel is inclined at an angle β = 30° with the horizontal direction.

Figure 1 Flow Domain



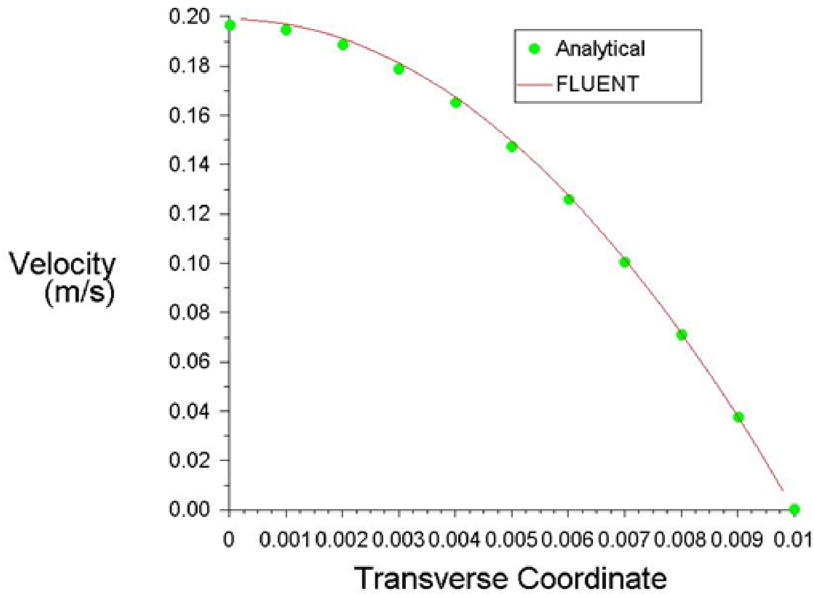
Material Properties	Geometry	Boundary Conditions
Density = 800 kg/m^3 Viscosity = 1 kg/m-s	Dimensions of the domain: 1 m X 18 m Angle with X-axis = 30°	Gauge Pressure at Inlet = 0 N/m^2 Gauge Pressure at Outlet = -706.32 N/m^2

Analysis Assumptions and Modeling Notes

The density based solver is used in ANSYS FLUENT. Pressure boundaries are specified to model the driving head in the direction of flow.

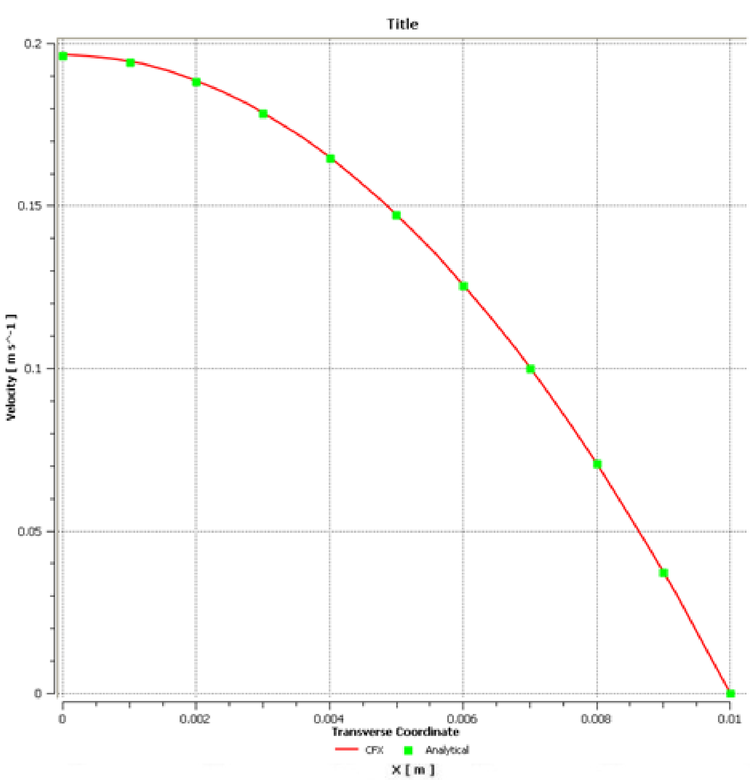
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Velocity Profile at Outlet



Results Comparison for ANSYS CFX

Figure 3 Comparison of Velocity Profile at Outlet



VMFL039: Boiling in a Pipe with Heated Wall

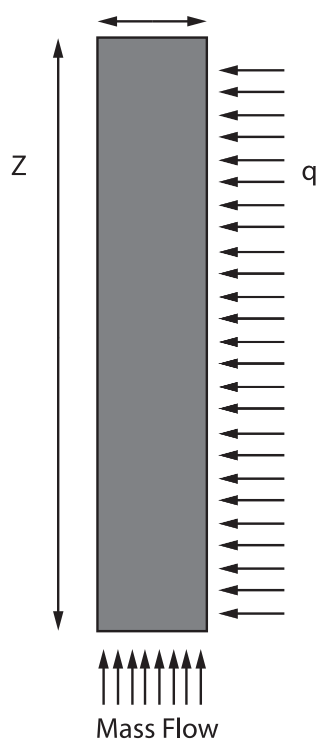
Overview

Reference	G.G. Bartolomej, V.G. Brantov, Y.S. Molochnikov, Y.V. Kharitonov, V.A. Solodkii, G.N. Batashova, V.N. Mikhailov. "*An experimental investigation of the true volumetric vapour content with subcooled boiling tubes*". *Thermal Engineering*. Vol. 29, No. 3. 20-22. 1982.
Solver	ANSYS CFX (ANSYS FLUENT simulation is not available for this case)
Physics/Models	Multiphase flow, phase change, RPI Wall boiling Model
Input File	`wall-boiling.def`

Test Case

Bubble formation and boiling near the heated wall of a vertical pipe are modeled. Outer wall of the pipe is heated with a constant heat flux.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Steam-Water 2-phase Flow: • Water: continuous phase	Radius of the pipe = 7.7 mm Height of the pipe = 2 m	Mass flux at inlet = 900 $kg/m^2/s$ Inlet pressure = 4.5 X 10^6 N/m^2

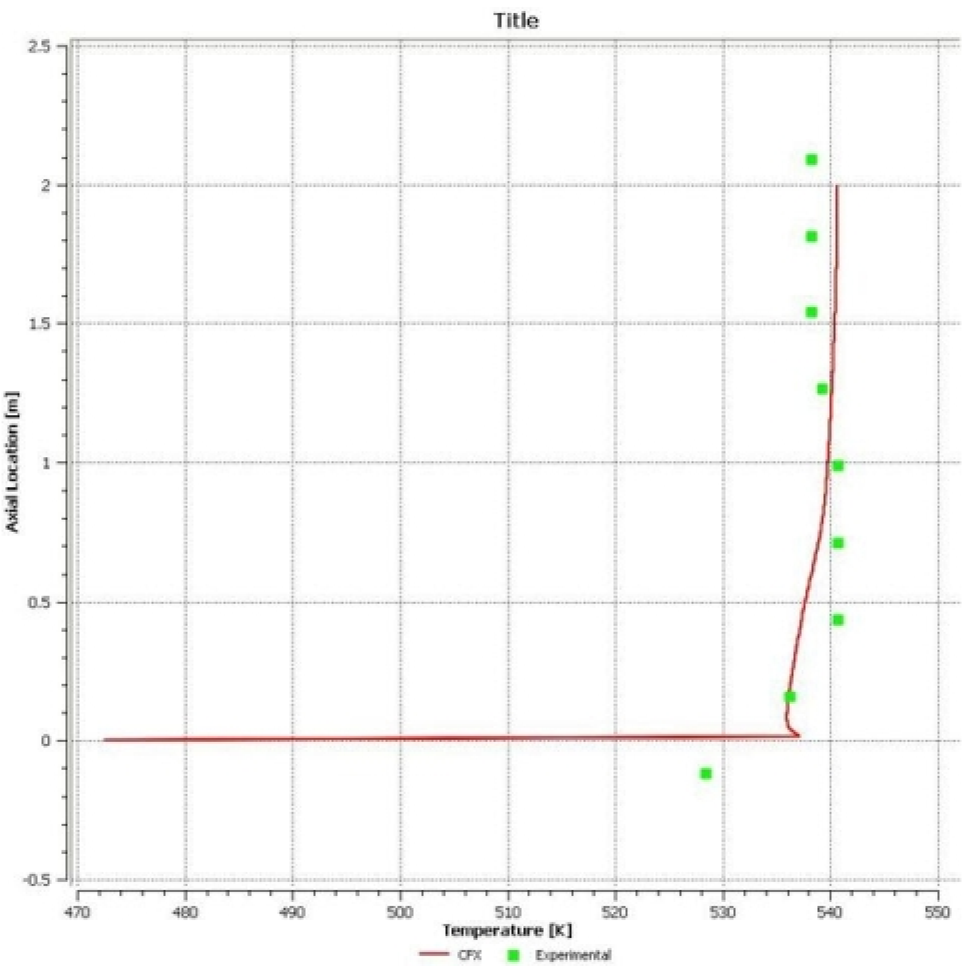
Material Properties	Geometry	Boundary Conditions
• Water Steam: dispersed bubbles Bubble diameter dependent on fluid temperature		Heat transfer at the wall = 570000 W/m^2

Analysis Assumptions and Modeling Notes

The flow is steady. SST model is used for turbulence. RPI model for wall boiling is used with a value of 0.8 for the wall area fraction affected by vapor.

Results Comparison

Figure 2 Comparison of Temperature Along the Pipe Wall



VMFL040: Separated Turbulent Flow in Diffuser

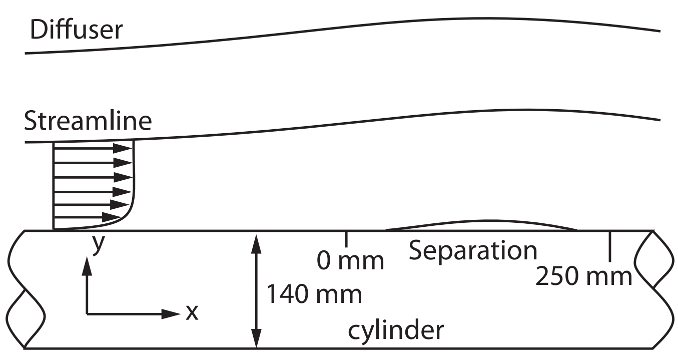
Overview

Reference	D.M. Driver. "*Reynolds shear stress measurements in a separated boundary layer flow*". *AIAA-91-1787*. 1991.
Solver	ANSYS CFX, ANSYS FLUENT
Physics/Models	SST model, Adverse pressure gradient, flow separation
Input Files	`diffuser-sep.def` for ANSYS CFX `VMFL040A_diffuser-sep.cas` for ANSYS FLUENT

Test Case

The test case geometry is shown in *Figure 1* (p. 145). It consists of an axisymmetric diffuser with an internally mounted cylinder along the centre line. The curvature of the diffuser wall results in an adverse pressure gradient. A relatively short separation region was detected in the experiment.

Figure 1 Sketch of Flow Domain



This figure is not to scale.

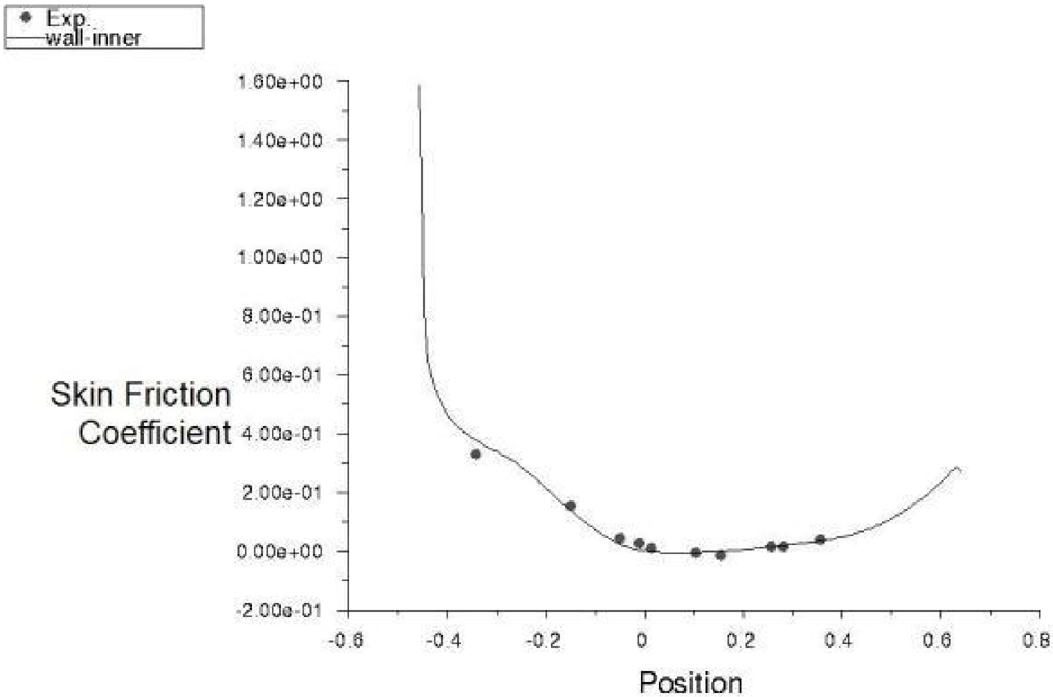
Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 1.5 X 10^{-5} kg/m-s	Diameter of the cylinder = 140 mm Length of the domain = 1100 mm	Velocity profile at inlet with average velocity = 29 m/s Outer wall modeled as a slip (inviscid) wall

Analysis Assumptions and Modeling Notes

The flow is steady. SST model is used for turbulence.

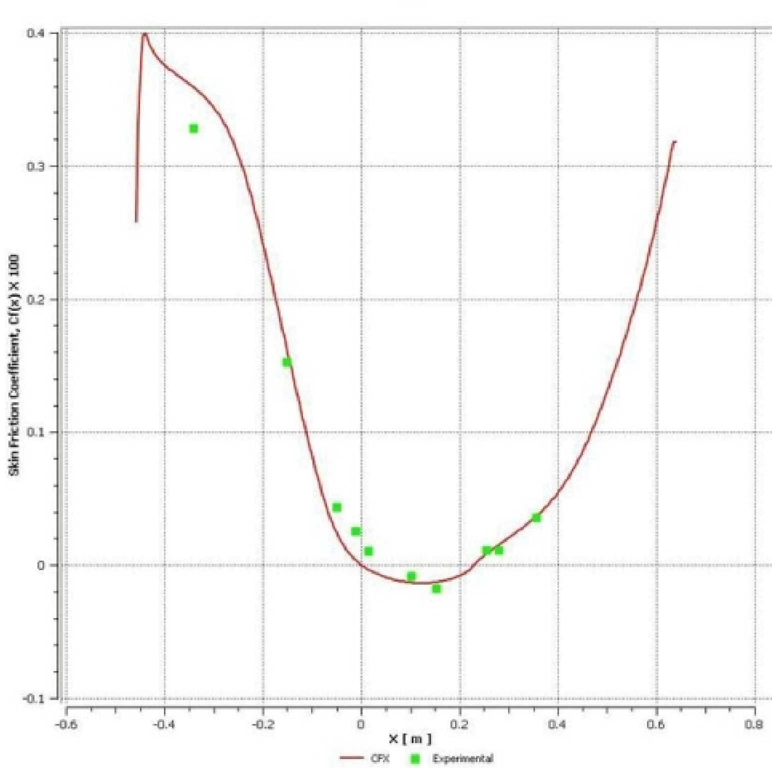
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Skin Friction Coefficient on the Cylinder Wall



Results Comparison for ANSYS CFX

Figure 3 Comparison of Skin Friction Coefficient on the Cylinder Wall



VMFL041: Transonic Flow Over an Airfoil

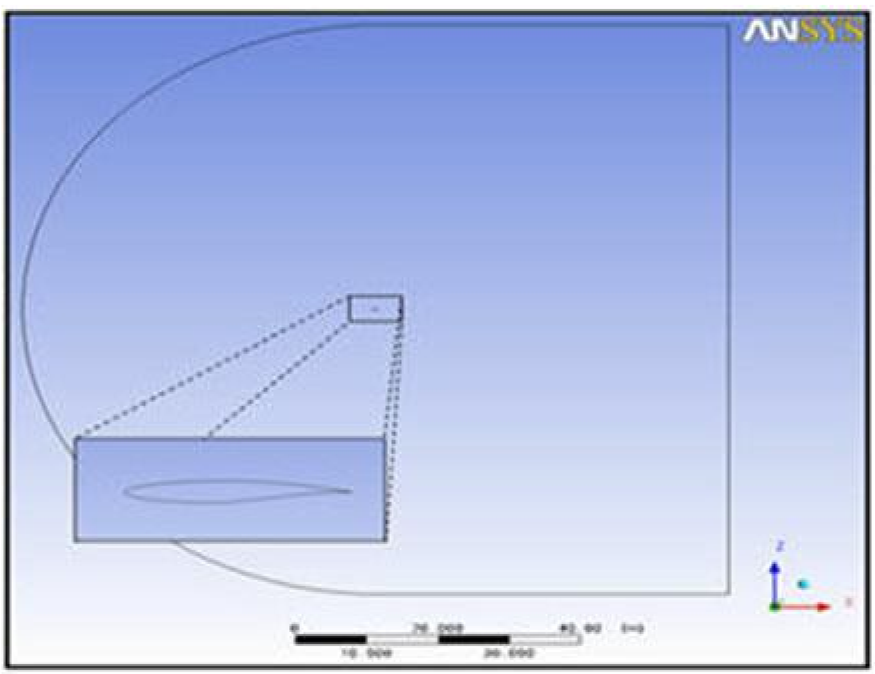
Overview

Reference	P.H. Cook, M.A. McDonald, and M.C.P. Firmin. "*AEROFOIL RAE 2822 - PRESSURE DISTRIBUTIONS, AND BOUNDARY LAYER AND WAKE MEASUREMENTS.*" *AGARD Advisory Report No. 138.*
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Transonic flow, shock, SST model
Input File	`VMFL041_transonic.cas` for ANSYS FLUENT `air_foil.def` for ANSYS CFX

Test Case

Transoinc flow over air foil RAE 2822 is modeled for an angle of attack of 3.19°. The flow domain spans over 100 Chord lengths in both stream-wise and transverse directions.

Figure 1 Flow Domain



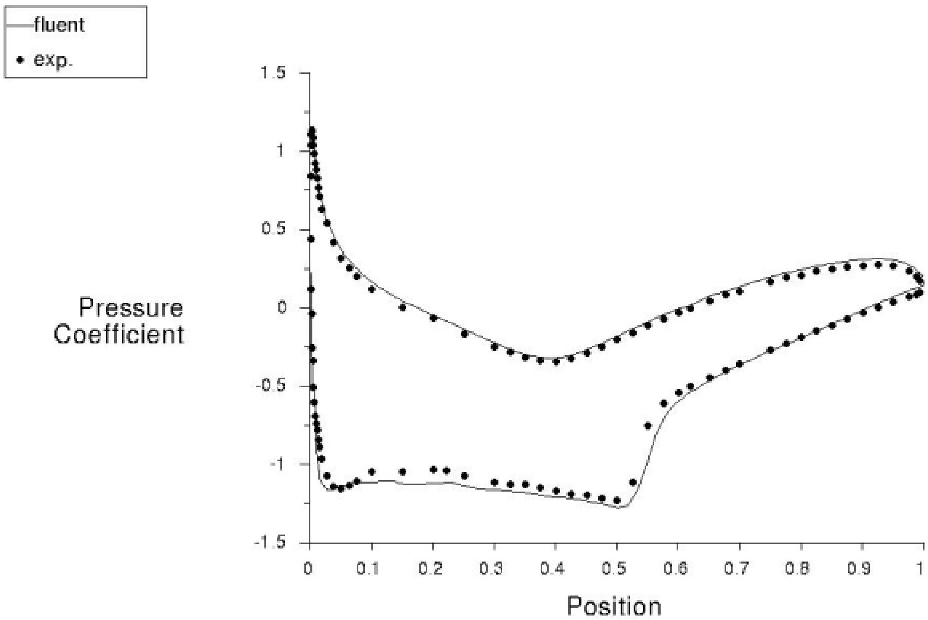
Material Properties	Geometry	Boundary Conditions
Density: Ideal gas law for Air Viscosity: 1.831 X 10^{-5} kg/m-s	Chord length of the airfoil = 1 m	Velocity profile at inlet with an average velocity of 218 m/s

Analysis Assumptions and Modeling Notes

The flow is steady. The inlet flow Mach number is close to transonic range. Walls are assumed to be adiabatic.

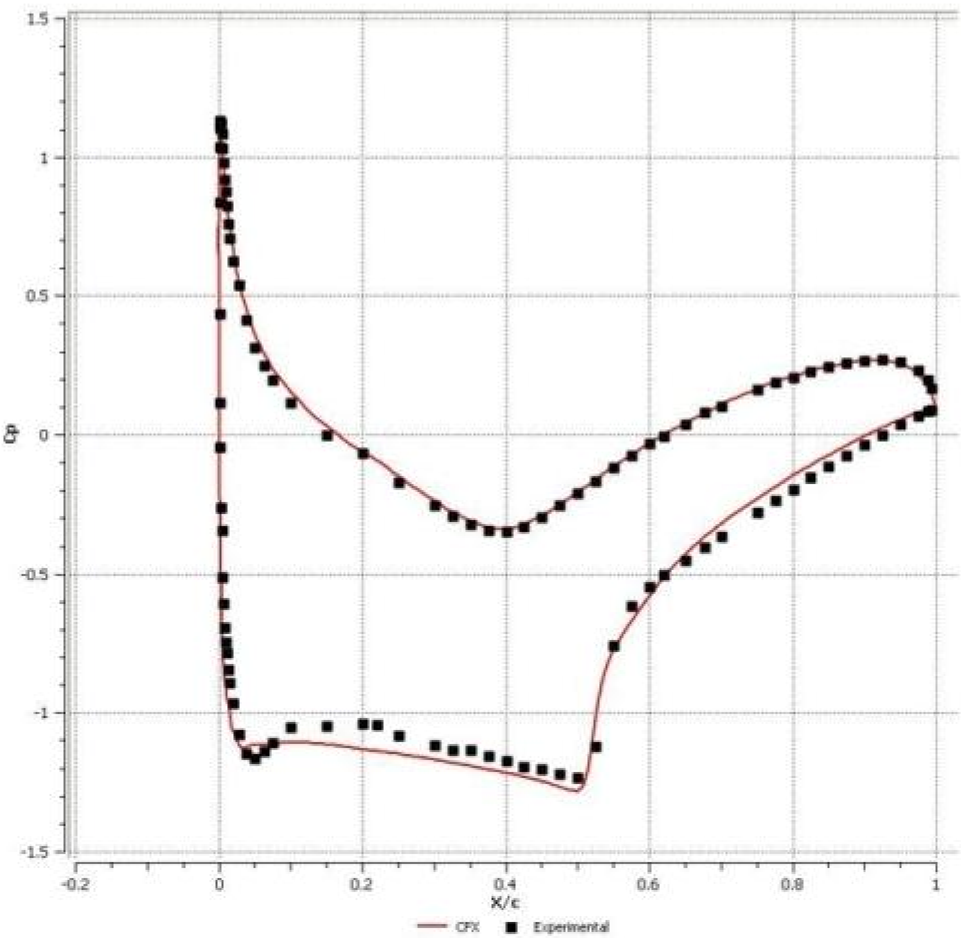
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Pressure Coefficient on the Airfoil



Results Comparison for ANSYS CFX

Figure 3 Comparison of Pressure Coefficient on the Airfoil



VMFL042: Turbulent Mixing of Two Streams with Different Density

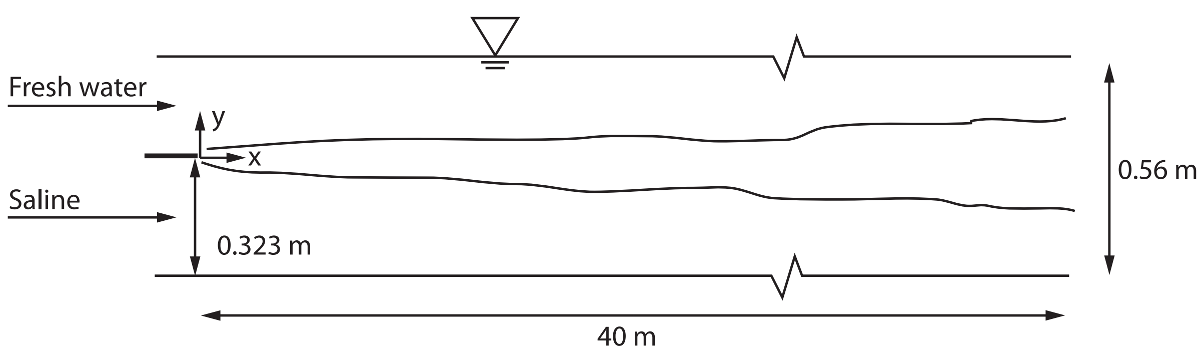
Overview

Reference	R.E. Uittenbogaard. "*Stably Stratified Mixing Layer*". *Data Report for the 14th meeting of the IAHR Working Group on Refined Flow Modeling*. 1989. R.E. Uittenbogaard. "*The Importance of Internal Waves for Mixing in a Stratified Estaurine Tidal Flow*". 1995.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	SST model, mixing layer, density difference, buoyancy
Input File	`VMFL042_mixing.cas` for ANSYS FLUENT `saline-mixing_layer.def` for ANSYS CFX

Test Case

Mixing of two turbulent streams of fresh water and saline water is modeled. The two streams are parallel at the inlet and mixing proceeds downstream.

Figure 1 Flow Domain



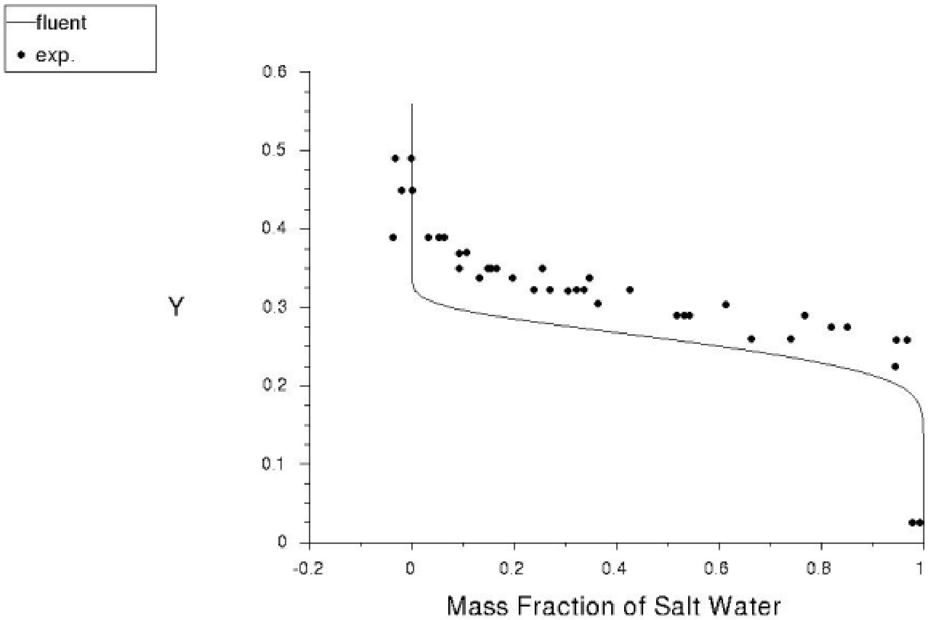
Material Properties	Geometry	Boundary Conditions
Density of fresh water: 1015 kg/m^3 Density of saline water: 1030 kg/m^3 Mixture kinematic diffusivity: 1×10^{-9} m^2/s	Length of the mixing duct = 40 m	Fresh water inlet velocity = 0.52 m/s Salt water inlet velocity = 0.32 m/s

Analysis Assumptions and Modeling Notes

The flow is steady. SST model is used. Buoyancy turbulence production option is used.

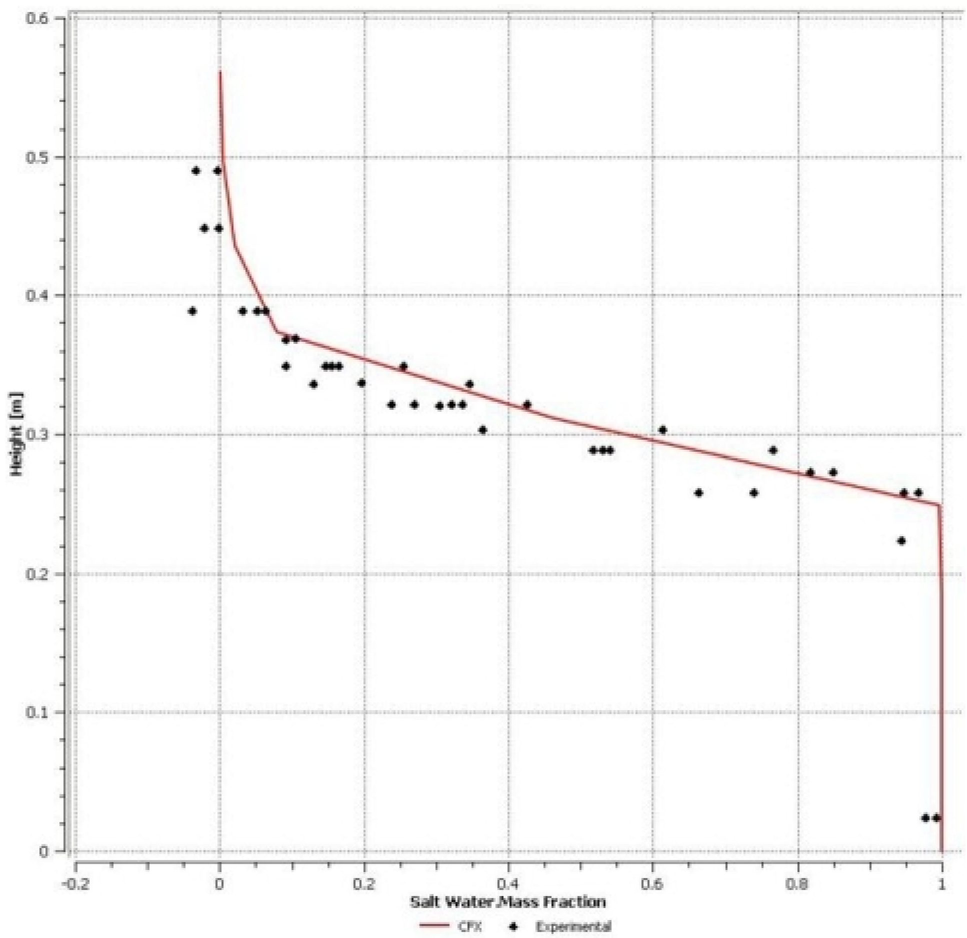
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Mass Fraction of Salt Water Across the Mixing Layer at x = 10m



Results Comparison for ANSYS CFX

Figure 3 Comparison of Mass Fraction of Salt Water Across the Mixing Layer at x = 10m



VMFL043: Laminar to Turbulent Transition of Boundary Layer over a Flat Plate

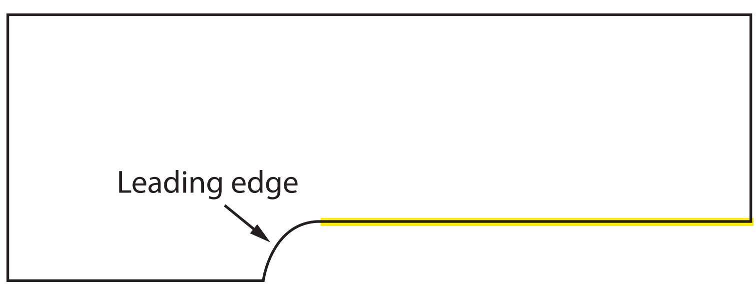
Overview

Reference	A. M. Savill. "*Some recent progress in the turbulence modeling of bypass transition*". *Near-Wall Turbulent Flows* (eds. R. M. C. So, C. G. Speziale & B. E. Launder). Elsevier Science Publishers. 829-848. 1993. P.E. Roach, and D.H. Brierley. "*The influence of a turbulent free stream on zero pressure gradient transitional boundary layer development. Part I: Test Cases T3A and T3B*". Numerical *Simulation of Unsteady and Transition to Turbulence* (eds. Pironneau, Rodi, Ryhming, Savill, and Truong). Cambridge University Press. Cambridge. 319-347. 1992.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	SST model, transitional flow
Input File	**`VMFL043_transition.cas`** for ANSYS FLUENT **`bl-transition.def`** for ANSYS CFX

Test Case

Laminar to turbulent transition of a boundary layer over a flat plate is modeled. The free stream turbulence intensity is 3.3%.

Figure 1 Flow Domain



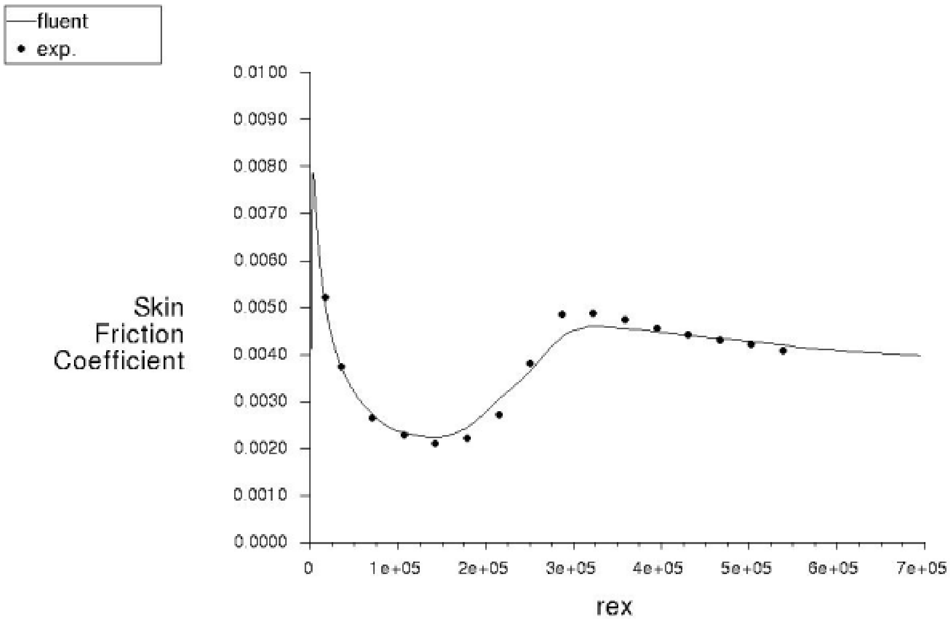
Material Properties	Geometry	Boundary Conditions
Density: 1.2 kg/m^3 Viscosity: 1.831 X 10^{-5} kg/m-s	Length of the flat plate = 2m	Inlet Velocity = 5.3 m/s Inlet eddy viscosity ratio = 9.7

Analysis Assumptions and Modeling Notes

The flow is steady. SST model with Gamma Theta model for transitional turbulence is used. Langry Menter correlation was used for transition onset.

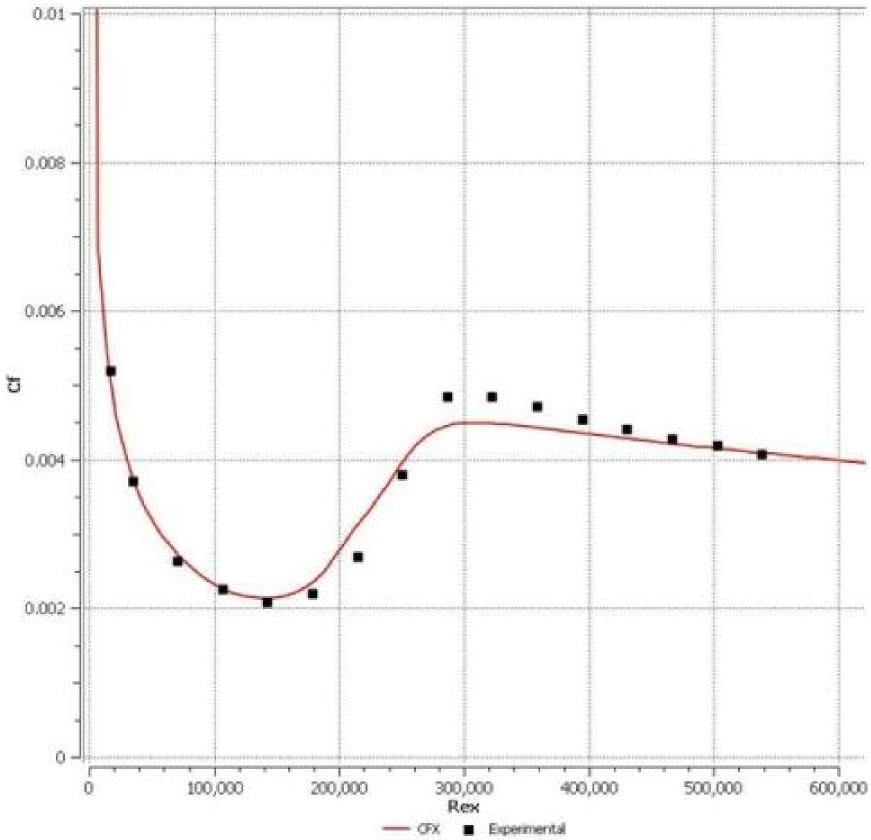
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Skin Friction Coefficient on the Plate



Results Comparison for ANSYS CFX

Figure 3 Comparison of Skin Friction Coefficient on the Plate



VMFL044: Supersonic Nozzle Flow

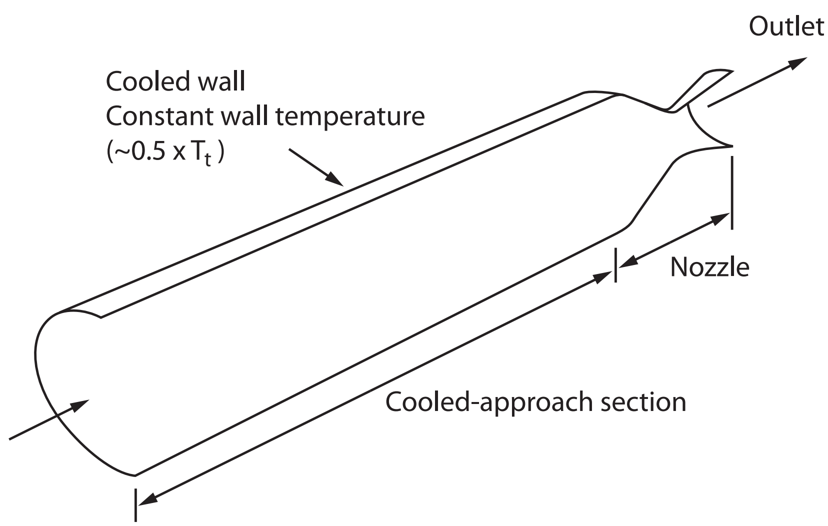
Overview

Reference	L.H. Back, P.F. Massier, and H.L. Gier. "*Convective Heat Transfer in a Convergent-Divergent Nozzle*". *Int. J. Heat Mass Transfer*. Vol. 7. 549-568. 1964.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Compressible flow in supersonic regime, SST Model
Input File	`VMFL044_nozzleflow.cas` for ANSYS FLUENT `supersonic-nozzle.def` for ANSYS CFX

Test Case

Supersonic flow in a convergent-divergent nozzle is modeled. The flow is supersonic in the entire divergent section of the nozzle.

Figure 1 Flow Domain



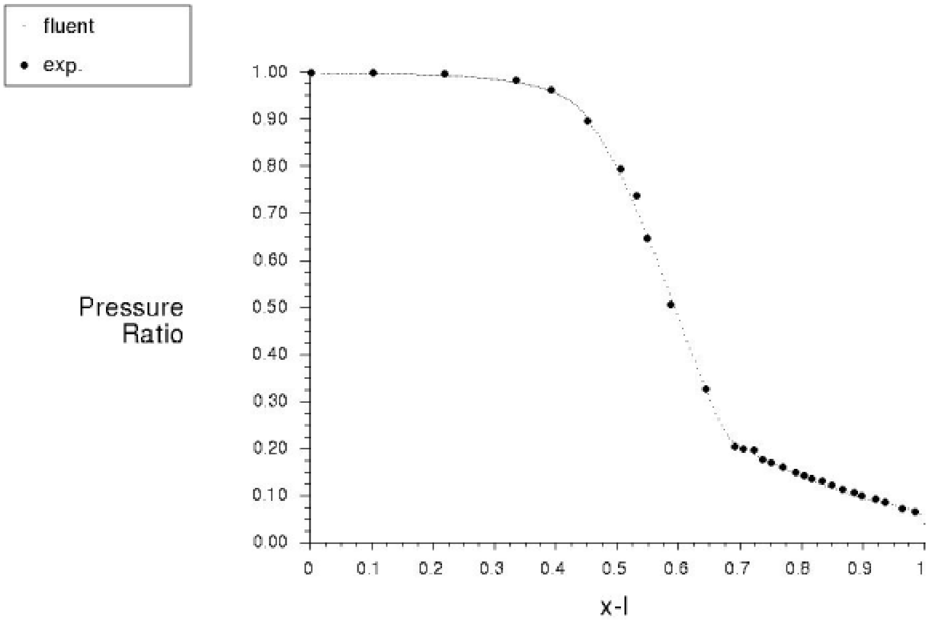
Material Properties	Geometry	Boundary Conditions
Density: Ideal Gas Viscosity: 1.831 X 10^{-5} kg/m-s	Length of the nozzle = 0.1594 m Exit-to-throat area ratio = 2.68 Half angle of divergence = 15°	Inlet Relative Pressure = 1 X 10^6 Pa Inlet Total Temperature = 825 K Wall temperature = 413 K

Analysis Assumptions and Modeling Notes

The flow is steady. The walls are assumed to be at constant temperature. Only a 3° sector of the domain is modeled due to symmetry.

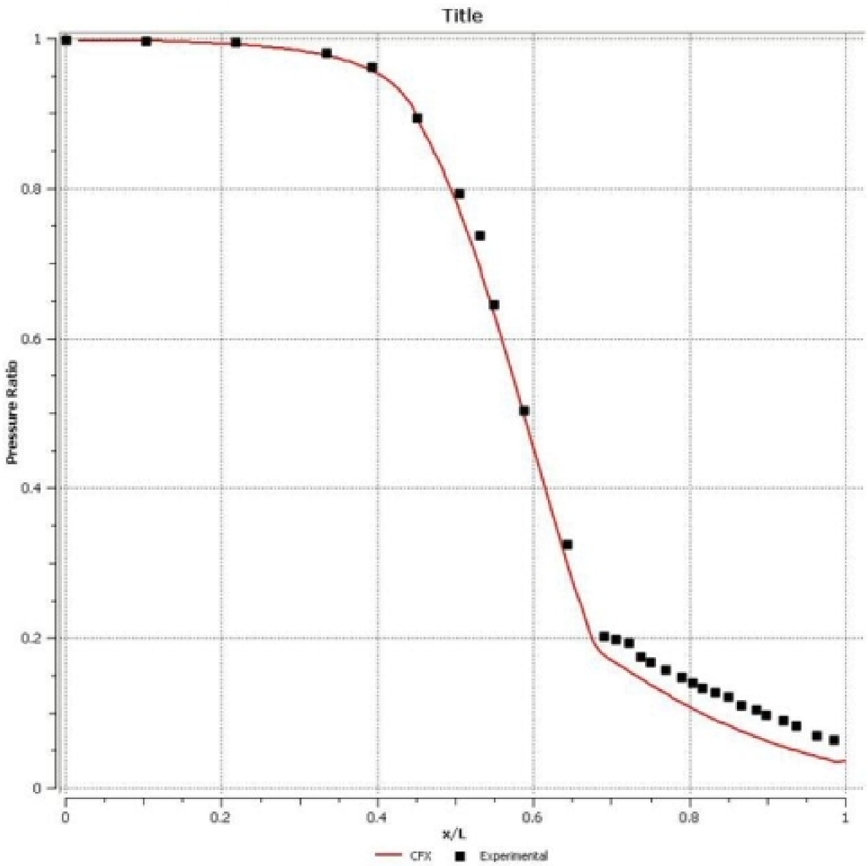
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Pressure Ratio Along the Nozzle Wall with Experimental Data



Results Comparison for ANSYS CFX

Figure 3 Comparison of Pressure Ratio Along the Nozzle Wall with Experimental Data



VMFL045: Oblique Shock over an Inclined Ramp

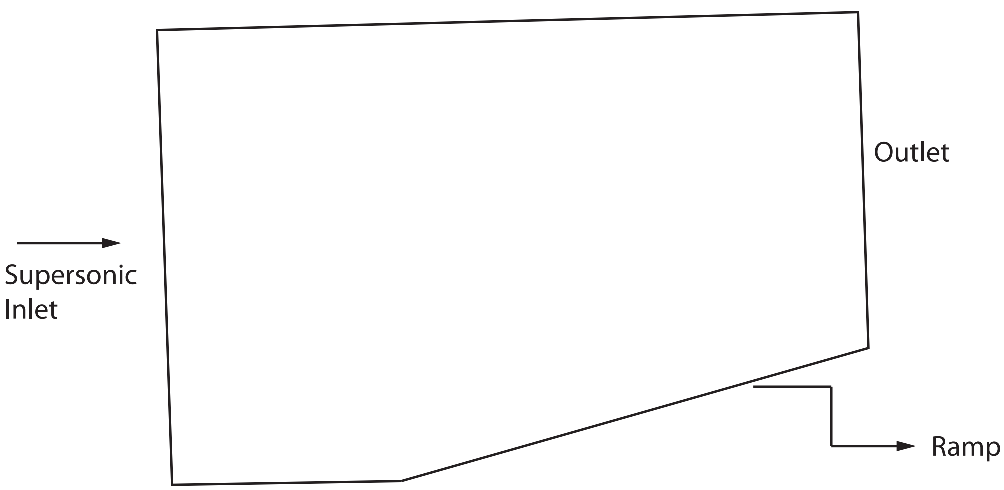
Overview

Reference	F. M. White. "*Fluid Mechanics*". 3rd Edition. McGraw-Hill Book Co.. New York, NY. 560-567. 1994.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Compressible flow in supersonic regime, Oblique shock
Input File	`VMFL045_obliqueshock.cas` for ANSYS FLUENT `ramp_supersonic_tet.def` for ANSYS CFX

Test Case

Supersonic flow over a 15° ramp is modeled. The ramp leads to the formation of an oblique shock. Inlet Mach number is about 2.5. The flow is laminar. The ANSYS CFX values are taken at a Point 1 (x=0.38 m, y=0.14 m)

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density: Ideal Gas Viscosity: 1 X 10^{-8} kg/m-s	Angle of the ramp = 15° Width of the domain (in transverse direction) = 0.3048 m	Inlet velocity = 852.68 m/s Inlet Temperature = 289 K Wall: Adiabatic

Analysis Assumptions and Modeling Notes

The flow is steady and laminar. The walls are assumed to be adiabatic.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Properties Downstream of the Oblique Shock

	Target	ANSYS FLUENT	Ratio
Mach Number	1.87	1.9	1.01
Temperature	382 K	377.6 K	0.99
Density	2.279 kg/m^3	2.233 kg/m^3	0.98

Results Comparison for ANSYS CFX

Table 2 Comparison of Properties Downstream of the Oblique Shock

	Target	ANSYS CFX	Ratio
Mach Number	1.87	1.88	1.00
Temperature	382 K	381.6 K	0.999
Density	2.279 kg/m^3	2.30 kg/m^3	1.01

VMFL046: Supersonic Flow with Normal Shock in a Converging Diverging Nozzle

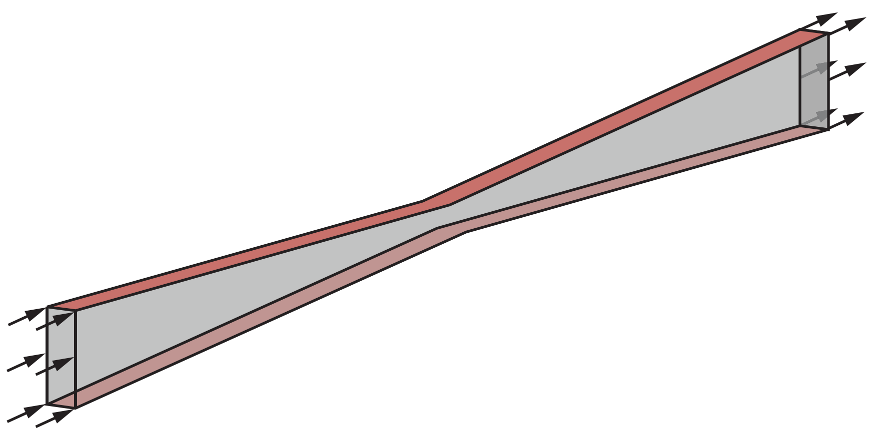
Overview

Reference	F. M. White. "*Fluid Mechanics*". 3rd Edition. McGraw-Hill Book Co.. New York, NY. 518-531. 1994.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Compressible flow in supersonic regime, Normal shock
Input File	`VMFL046_supersonic.cas` for ANSYS FLUENT `Supersonic2x.def` for ANSYS CFX

Test Case

Supersonic flow in a CD nozzle is modeled. The maximum Mach number is 2.2. A normal shock is formed in the divergent section. Mach number distribution in the nozzle is compared with analytical solution for nozzle flow.

Figure 1 Flow domain



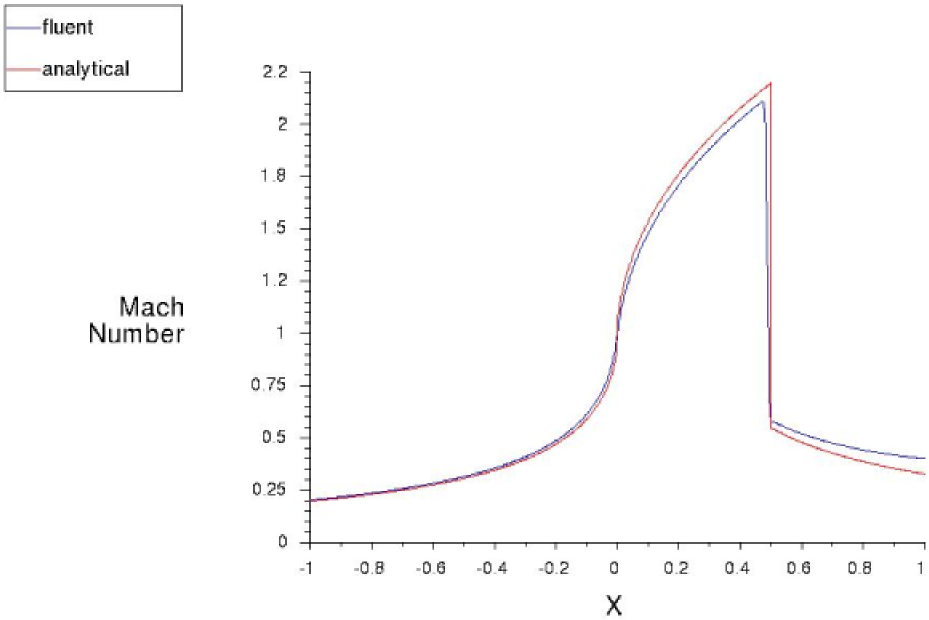
Material Properties	Geometry	Boundary Conditions
Density: Ideal Gas Viscosity: 1.831 X 10^{-5} kg/m-s	Length of the nozzle = 2m Exit-to-throat area ratio = 3	Inlet Relative Pressure 200 kPa Inlet Total Temperature = 500 K Wall temperature = 328 K Outlet Relative Pressure (gauge) = 75 kPa

Analysis Assumptions and Modeling Notes

The flow is steady. The walls are assumed to be adiabatic. The flow is modeled as laminar.

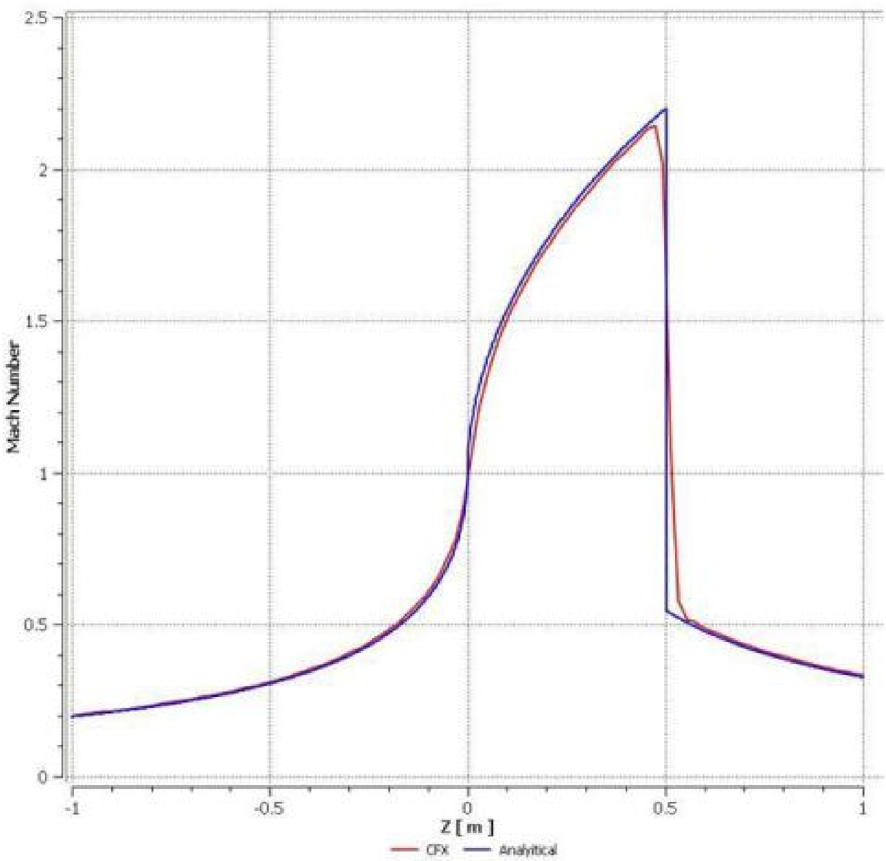
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Mach Number Along Center Line of the Nozzle With Analytical Solution



Results Comparison for ANSYS CFX

Figure 3 Comparison of Mach Number Along Center Line of the Nozzle With Analytical Solution



VMFL047: Turbulent Flow with Separation in an Asymmetric Diffuser

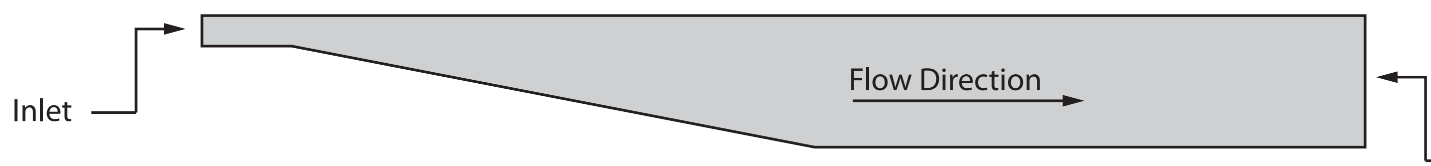
Overview

Reference	C. U. Buice and J. K. Eaton. "*Experimental Investigation of Flow Through an Asymmetric Plane Diffuser*". *J. Fluids Engineering*. Volume 122 (June 2000): 433-435.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent separation, standard k-ω model in ANSYS FLUENT, and SST model in ANSYS CFX.
Input File	`VMFL047_FLUENT.cas` for ANSYS FLUENT `VMFL047_CFX.def` for ANSYS CFX

Test Case

Turbulent flow with gradual separation and reattachment is modeled in an asymmetric 2-D diffuser. The lower wall of the diffuser is divergent with an angle of 10° and expands to about 4.7 times the inlet height.

Figure 1 Flow Domain



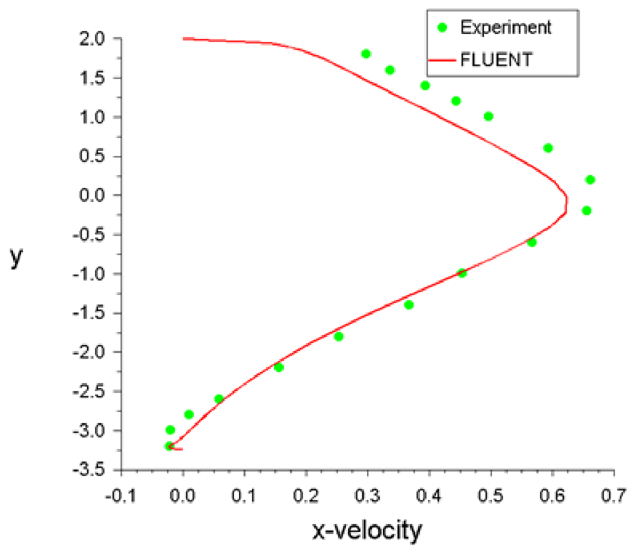
Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 0.0001 kg/m-s	Inlet height H = 2 M Outlet height = 4.7 H Angle of the divergent section = 10° Length of the straight section after divergence = 21 H	Fully developed turbulent profile for velocity at inlet with an average velocity = 0.7041 m/s

Analysis Assumptions and Modeling Notes

Steady turbulent flow.

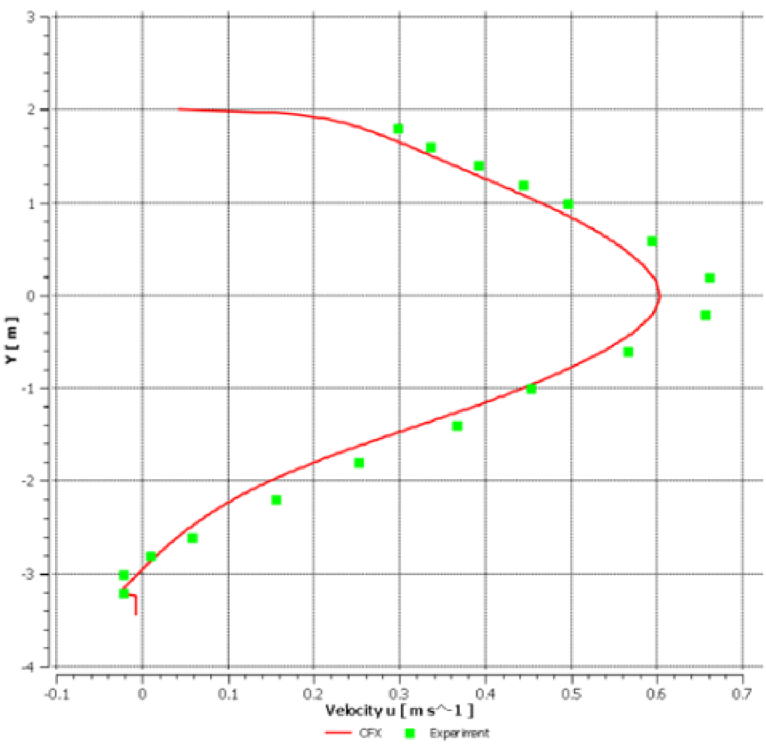
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of X-Velocity at X = 24.4 m



Results Comparison for ANSYS CFX

Figure 3 Comparison of X-Velocity at X = 24.4 m



VMFL048: Turbulent flow in a 180° Pipe Bend

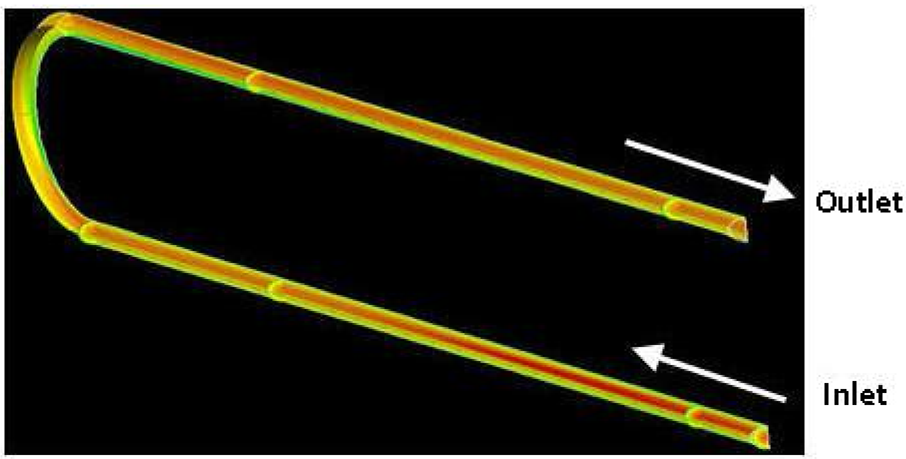
Overview

Reference	T. Takamasa and A. Tomiyama. "*Three-dimensional gas-liquid two-phase bubbly flow in a C-shaped tube*". *NURETH-9*. San Francisco, USA. 1-17. 1999.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	SST model, turbulent flow with separation and reattachment
Input File	`VMFL048_pipebend.cas` for ANSYS FLUENT `180deg_pipe_bend.def` for ANSYS CFX

Test Case

Flow in a 3-D pipe bend as shown in *Figure 1* (p. 173).

Figure 1 Flow Domain



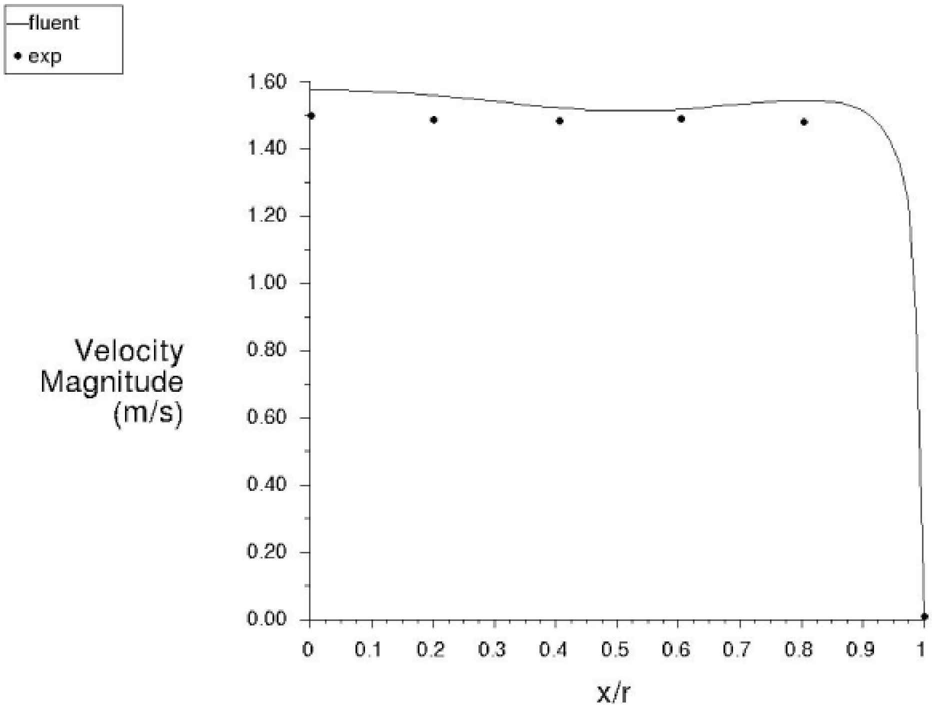
Material Properties	Geometry	Boundary Conditions
Density: 997 kg/m^3 Viscosity: 8.899 X 10^{-4} kg/m-s	Radius of the pipe = 14 mm Radius of the pipe bend = 125 mm	Velocity profile at inlet with an average velocity of 1.42 m/s

Analysis Assumptions and Modeling Notes

The flow is steady. Symmetry condition is applied on one side of the pipe.

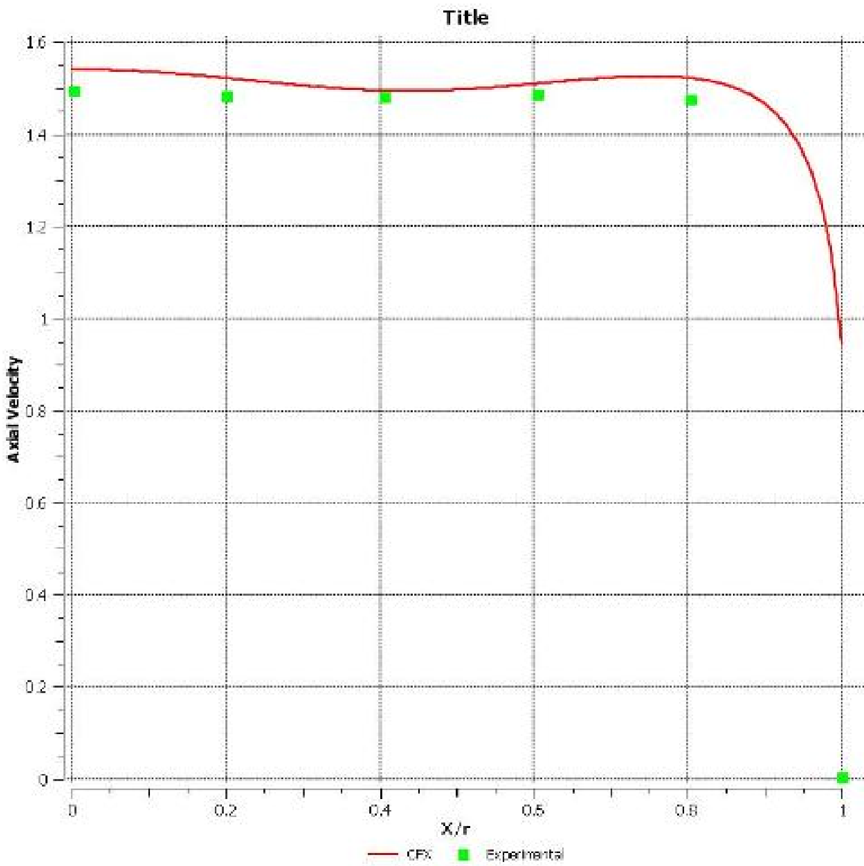
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Velocity in the Axial Direction at a Section 1.555 m upstream of the Outlet (after the bend)



Results Comparison for ANSYS CFX

Figure 3 Comparison of Velocity in the Axial Direction at a Section 1.555 m upstream of the Outlet (after the bend)



VMFL049: Combustion in an Axisymmetric Natural Gas Furnace

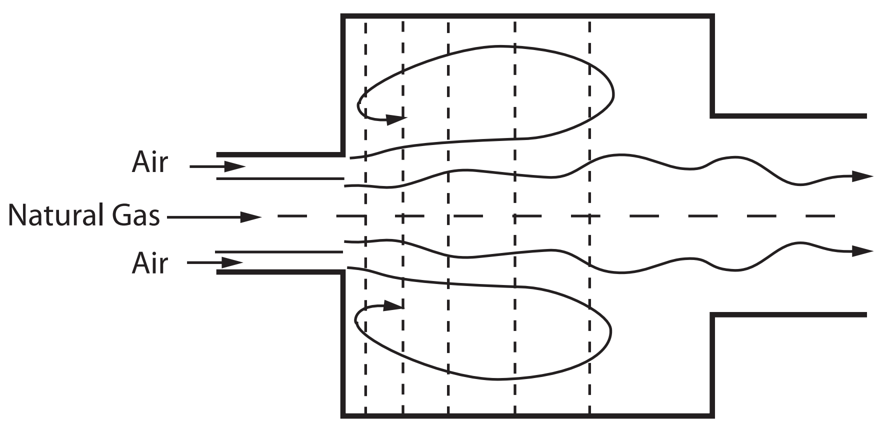
Overview

Reference	Westbrook and Dryer (1981), Coffee (1985).
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent non-premixed combustion, Eddy dissipation model, k-ε model
Target File	`VMFL049_combustion.cas` for ANSYS FLUENT `ENEL_Furnace.def` for ANSYS CFX

Test Case

Non-premixed combustion in a natural gas fired furnace is modeled. The axisymmetric flow field is modeled by a 3° cylindrical domain. Fuel jet consists of natural gas modeled as 93% Methane and 7% Nitrogen by mass.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Properties for the mixture of reactants used	Inner diameter of air annulus = 60 mm Outer diameter of air annulus = 100 mm Diameter of combustion chamber = 500 mm Length of chamber = 1700 mm	Air velocity at inlet = 34 m/s Fuel velocity at inlet = 4 m/s Wall temperature = 120°C CH_4 Mass fraction at inlet = 0.93

Analysis Assumptions and Modeling Notes

The flow is steady. Reactions modeled using Eddy Dissipation Model. The domain is axisymmetric.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of the Mole Fraction of CH_4 Along the Axis

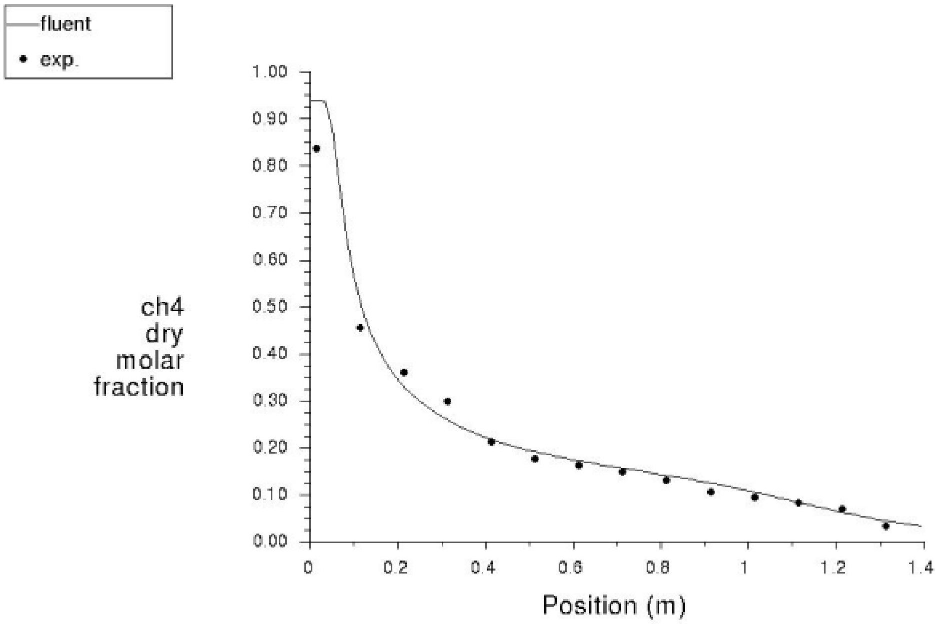
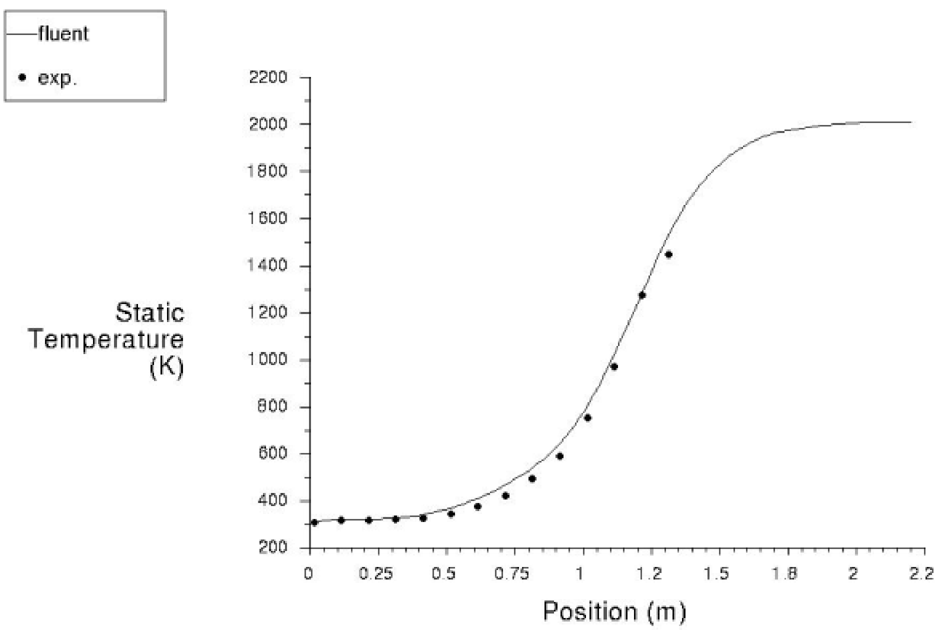


Figure 3 Comparison of Temperature Along the Axis



Results Comparison for ANSYS CFX

Figure 4 Comparison of the Mole Fraction of CH_4 Along the Axis

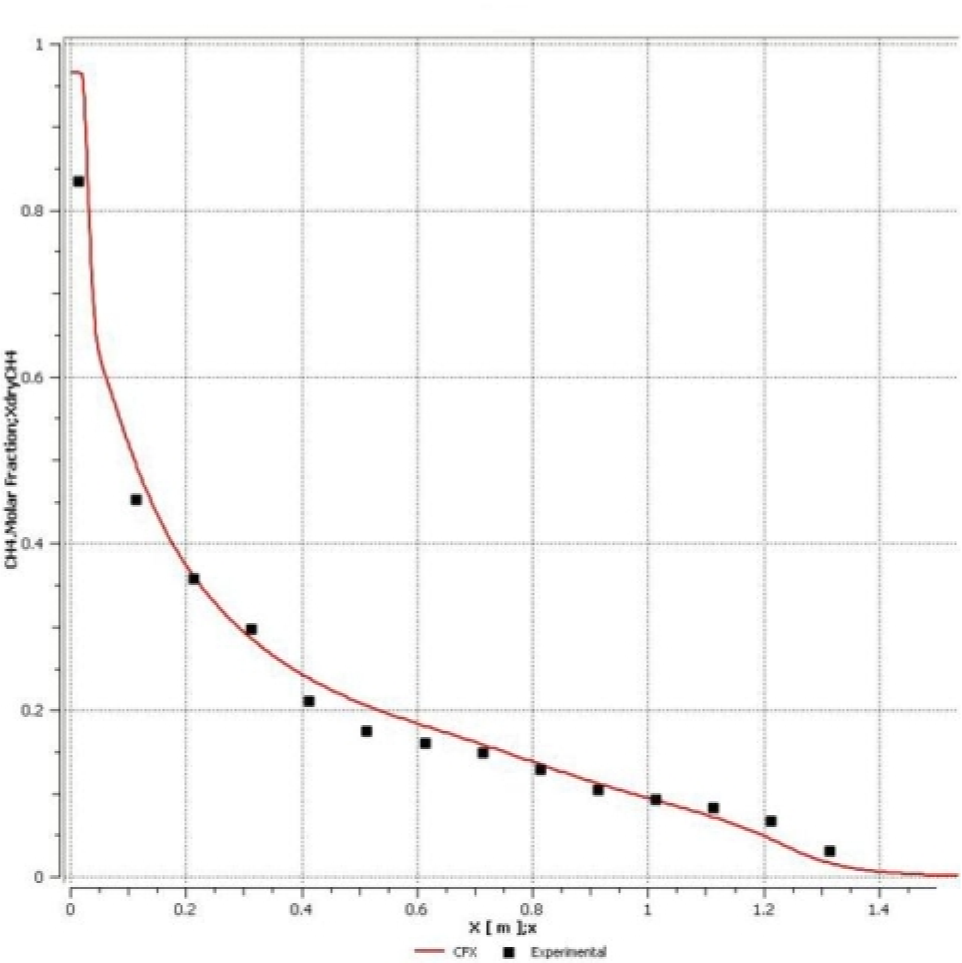
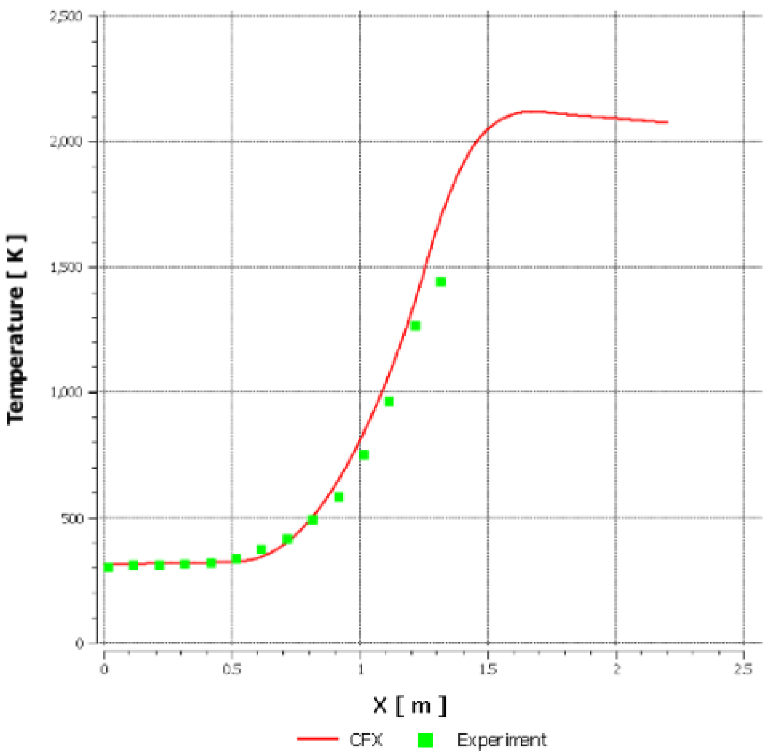


Figure 5 Comparison of Temperature Along the Axis



VMFL050: Transient Heat Conduction in a Semi-Infinite Slab

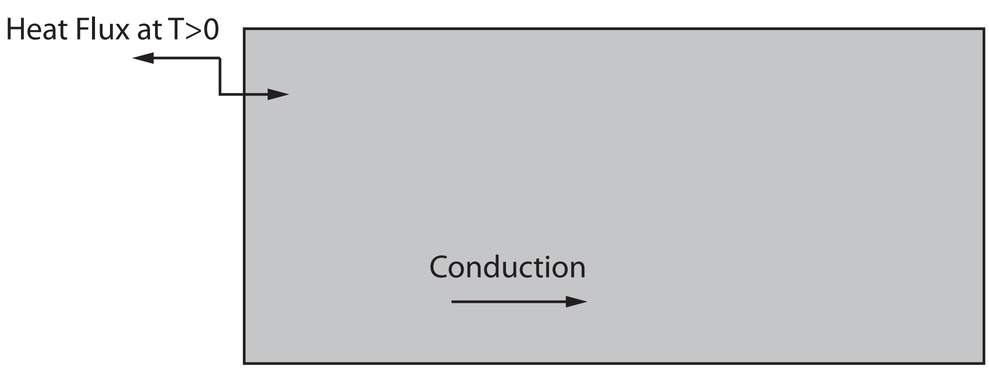
Overview

Reference	F.P. Incropera and D.P. Dewitt. "*Fundamentals of Heat and Mass Transfer*". 5th Edition. Wiley & Sons, 2002; Page 289.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Transient heat transfer, Conduction
Input File	**`VMFL050_FLUENT.cas`** for ANSYS FLUENT **`VMFL050_CFX.def`** for ANSYS CFX

Test Case

Unsteady heat conduction in a thick copper plate is modeled. Initially (at t = 0) the plate is at a uniform temperature of 293 K. It is suddenly exposed to a heat transfer at one surface with a constant heat flux of 3 X 10^5 W/m^2. The temperature distribution after 2 minutes is considered for verification.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density = 8995.67 kg/m^3 Specific Heat = 391 J/kg-K Conductivity = 401 W/m-K	Dimensions of the slab: 750 mm X 300 mm	Heat Flux = 3 X 10^5 W/m^2 on one wall. The opposite wall is adiabatic. Lateral boundaries are modeled as planes of symmetry.

Analysis Assumptions and Modeling Notes

The flow is steady transient. The dimensions considered her are adequate for the semi-infinite slab assumption. The domain is initialized with a uniform temperature of 293 K corresponding to the condition at time = 0 sec.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Temperature after 2 Minutes

	ANSYS FLUENT	Exact Solution	Ratio
Temperature of the all exposed to heat flux, at t=120 sec	392.95 K	393 K	0.9998
Temperature at a point 150 mm from the heat flux wall, at t=120 sec	318.41 K	318.4 K	1.0000

Results Comparison for ANSYS CFX

Table 2 Comparison of Temperature after 2 Minutes

	ANSYS CFX	Exact Solution	Ratio
Temperature of the wall exposed to heat flux, at t=120 sec	393 K	393 K	1.0000
Temperature at a point 150 mm from the heat flux wall, at t=120 sec	318.4 K	318.4 K	1.0000

VMFL051: Isentropic Expansion of Supersonic Flow over a Convex Corner

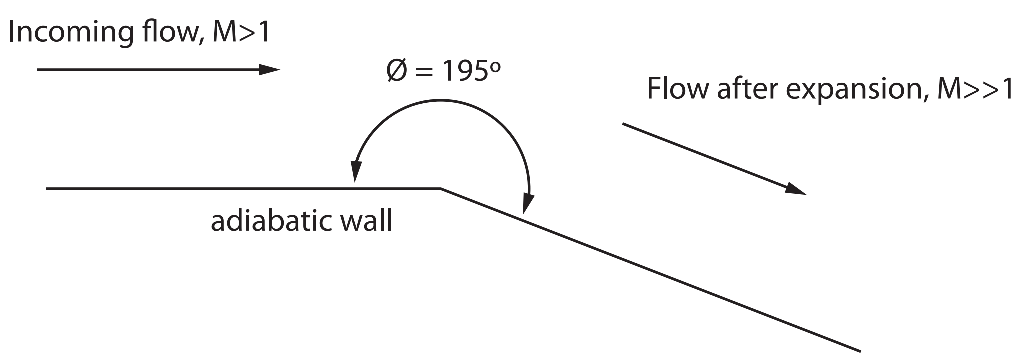
Overview

Reference	John Anderson. "*Modern Compressible Flow: With Historical Perspective*".
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Compressible, inviscid flow
Input File	**`VMFL051_FLUENT.cas`** for ANSYS FLUENT **`VMFL051_CFX.def`** for ANSYS CFX

Test Case

Centered expansion of inviscid supersonic flow around a corner is modeled. The expansion results in a change in direction of the flow, a drop in static pressure, and increase in Mach number. The approaching flow is supersonic, with a Mach number of 2.5. The expansion process is reversible and adiabatic.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density: Ideal Gas law Specific Heat = 1006.43 J/kg-K Molecular weight = 28.966	Angle round the corner = 195°	Inlet: Pressure = 202636.9 Pa Mach number = 2.5, Static temperature = 300 K (In CFX, the corresponding velocity is specified). Wall is adiabatic.

Analysis Assumptions and Modeling Notes

The flow is steady and compressible. Inviscid and incompressible. Analytic expressions for isentropic expansion can be used to calculate the Mach number downstream of the corner.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Mach Number Downstream of the Corner, after Expansion

	ANSYS FLUENT	Analytical Calculation	Ratio
Mach number after expansion	3.2316	3.2370	0.9980

Results Comparison for ANSYS CFX

Table 2 Comparison of Mach Number Downstream of the Corner, after Expansion

	ANSYS CFX	Analytical Calculation	Ratio
Mach number after expansion	3.2340	3.2370	0.9990

VMFL052: Turbulent Natural Convection inside a Tall Cavity

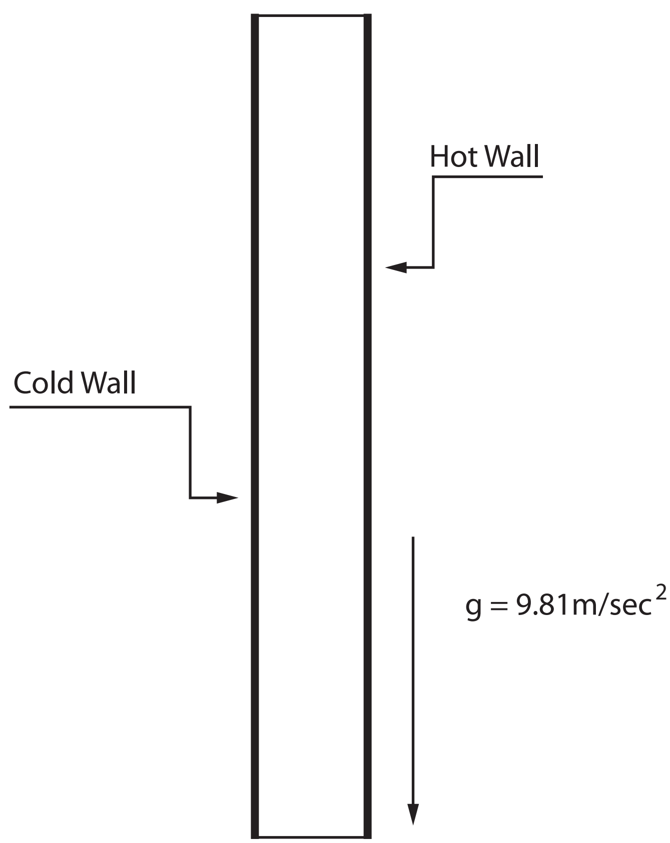
Overview

Reference	P.L. Betts and I.H. Bokhari. "*Experiments on turbulent natural convection in an enclosed tall cavity*". *International Journal of Heat and Fluid Flow*, 21. 675-683. 2000.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent flow, buoyancy effects, Boussinesq approximation (FLUENT)/Ideal gas (CFX)
Input File	`VMFL052_FLUENT.cas` for ANSYS FLUENT `VMFL052_CFX.def` for ANSYS CFX

Test Case

Natural convection in the turbulent flow field of an enclosed cavity with a length-to-width ratio of 28.6 is modeled. The Rayleigh number is in the turbulent range. The two vertical walls are kept at different temperatures, while the horizontal walls are adiabatic.

Figure 1 Flow Domain (not to scale)



Material Properties	Geometry	Boundary Conditions
Density : Boussinesq approximation (FLUENT), Ideal Gas law (CFX) Specific Heat = 1005 J/kg-K Viscosity = 1.81X 10^{-5} kg/m-sec Molecular weight = 28.966	Length of the cavity = 2.18 m Width of the cavity = 0.0762 m	Temperature of Cold wall = 288.25 K Temperature of Hot wall = 307.85 K Top and bottom walls are adiabatic

Analysis Assumptions and Modeling Notes

The flow is steady and is induced by natural convective heat transfer.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Vertical Velocity at Y/h = 0.05

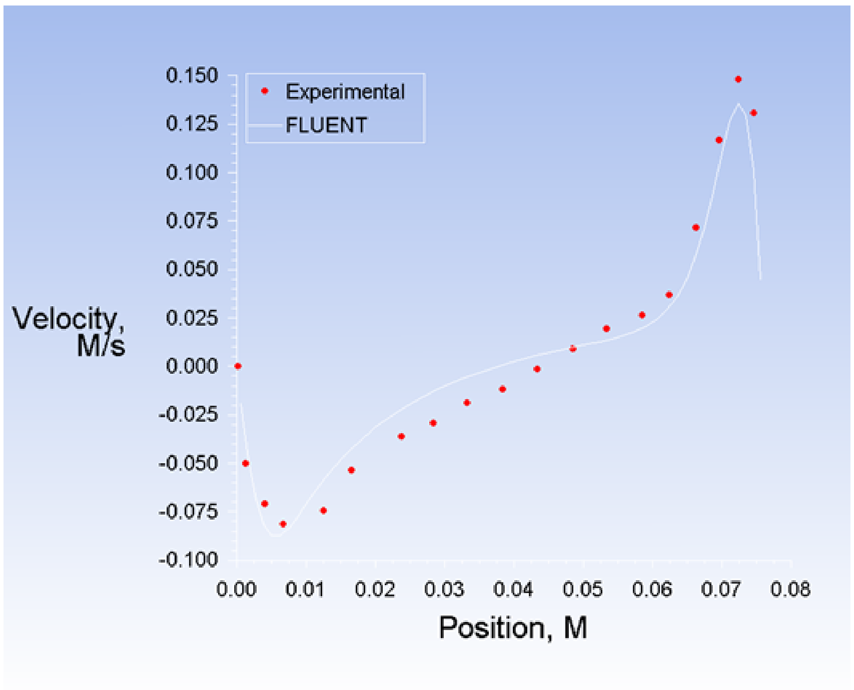
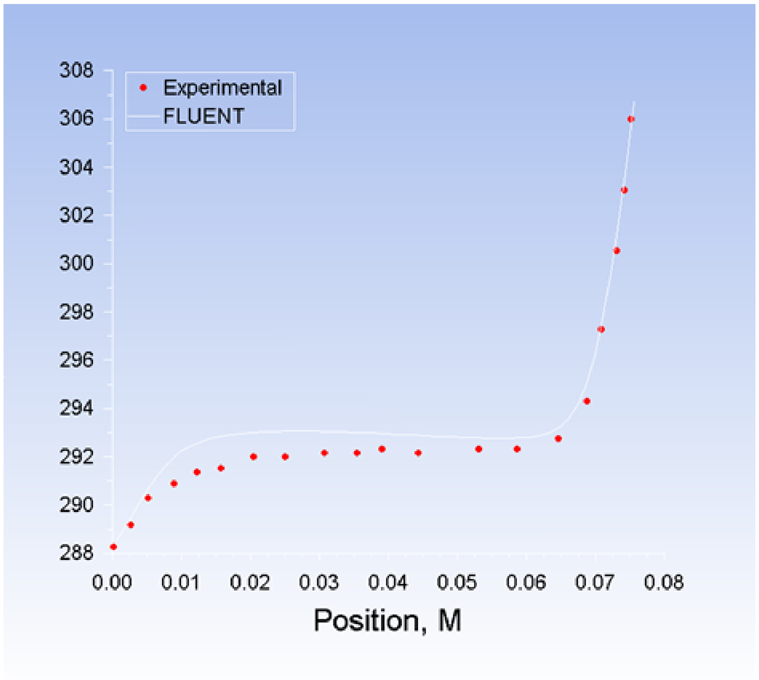


Figure 3 Comparison of Temperature at Y/h = 0.05



Results Comparison for ANSYS CFX

Figure 4 Comparison of Vertical Velocity at Y/h = 0.05

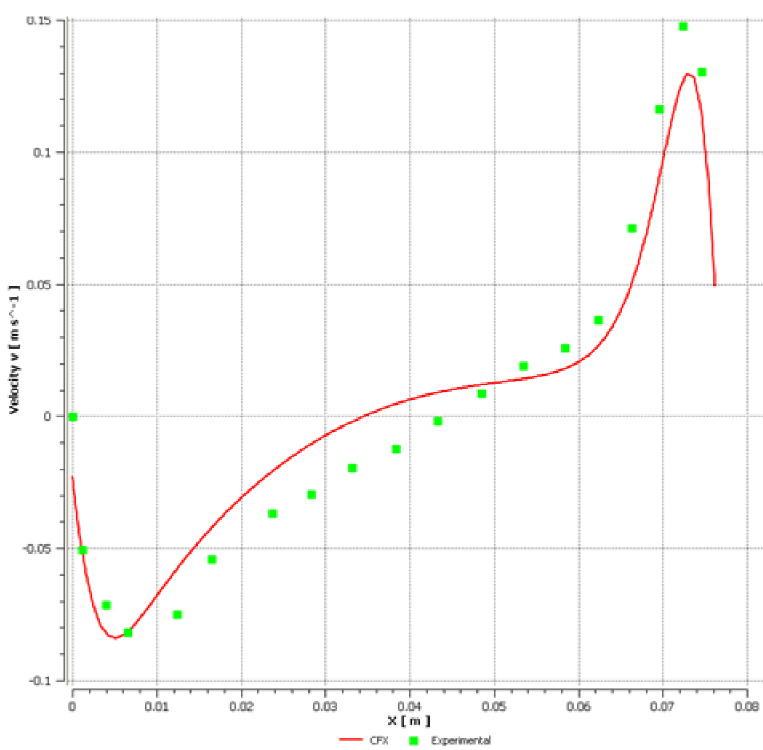
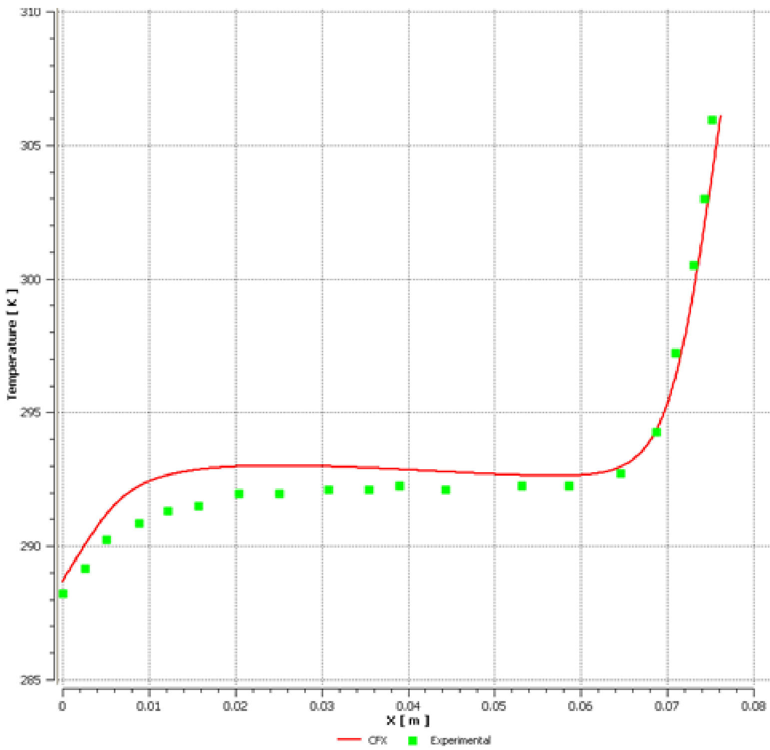


Figure 5 Comparison of Temperature at Y/h = 0.05



VMFL053: Compressible Turbulent Mixing Layer

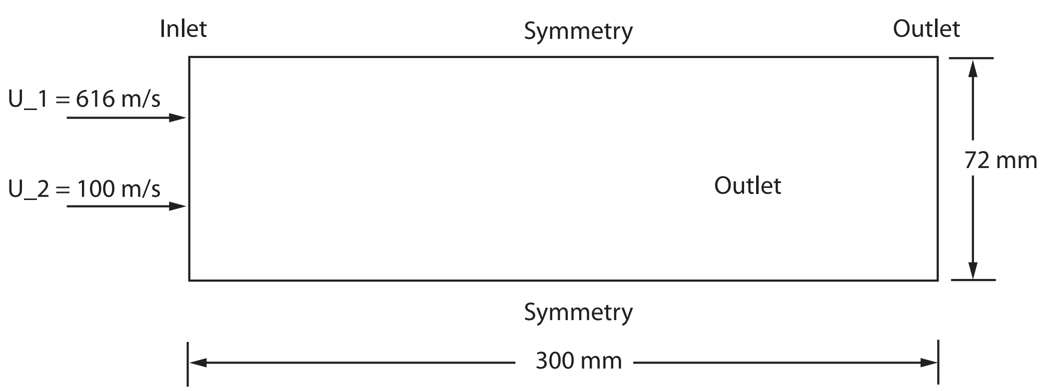
Overview

Reference	S.G. Goebel and J.C. Dutton. "*Experimental Study of Compressible Turbulent Mixing Layers*". *AIAA Journal*, 29(4). 538-546. 1991.
Solver	ANSYS FLUENT
Physics/Models	Turbulence: RNG k-ε model, compressible, energy equation
Input File	**`VMFL053_FLUENT.cas`** for ANSYS FLUENT

Test Case

Two streams of air are mixed in a rectangular tunnel. The length of the computational domain is chosen such that the local Reynolds number at the exit of the test section, which is based on the velocity difference between the streams and the mixing layer thickness, is greater than 100,000. This is the Reynolds number needed for the complete development of the mixing layer.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Air: Density: Ideal Gas Specific Heat: 1006.43 J/kg-K Thermal Conductivity: 0.0242 W/m-K Viscosity: 1.4399e-05 kg/m-s	Dimensions of the domain: 300mm X 72 mm	**Primary Stream (1):** Total Pressure = 487 kPa Static Pressure = 36 kPa Total Temperature = 360 K Mach Number = 2.35 Turbulent Kinetic Energy = 74 m^2/s^2 Turbulent Dissipation Rate = 62,300 m^2/s^3 **Secondary Stream (2):** Total Pressure = 38 kPa Static Pressure = 36 kPa Total Temperature = 290 K

Material Properties	Geometry	Boundary Conditions
		Mach Number = 0.36 Turbulent Kinetic Energy = 226 m^2/s^2 Turbulent Dissipation Rate = 332,000 m^2/s^3

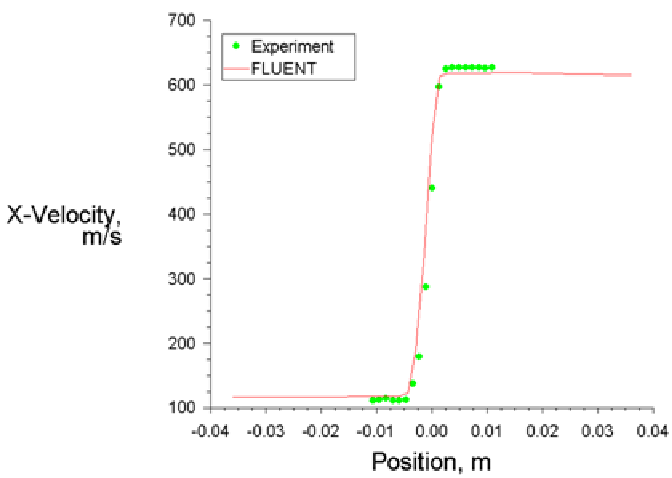
Analysis Assumptions and Modeling Notes

The flow is steady, turbulent, and compressible. The RNG k-ε model is used for turbulence.

Results Comparison

The velocity profiles as the mixing layer evolves are compared with the experimental data.

Figure 2 X Velocity Profiles at x = 50 mm



VMFL054: Laminar flow in a Trapezoidal Cavity

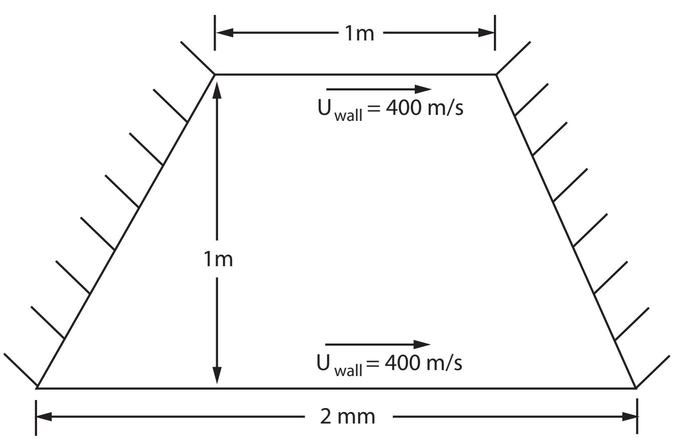
Overview

Reference	J.H. Darr and S.P. Vanka. "*Separated Flow in a Driven Trapezoidal Cavity*". *Phys. Fluids A*, 3(3):385-392. March 1991.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Viscous flow, driven by a moving walls
Input File	**`VMFL054_FLUENT.cas`** for ANSYS FLUENT **`VMFL054_CFX.def`** for ANSYS CFX

Test Case

Laminar flow induced by the motion of the top wall of a trapezoidal cavity. The top and bottom walls move but the side walls are stationary. The height of the cavity *h* is 1 m. The widths of the top and bottom walls are 1 m and 2 m, respectively (*Figure 1* (p. 191)).

Figure 1 Flow Domain



Material Properties	**Geometry**	**Boundary Conditions**
Density = 1 kg/m^3 Viscosity = 1 kg/m-s	Height of cavity = 1 m Width of the bottom base = 2 m Width of the top base = 1 m	Velocity of the base walls = 400 m/s Other walls are stationary

Analysis Assumptions and Modeling Notes

The flow is steady. A pressure based solver is used. A triangular mesh of 4016 cells is used to discretize the domain.

Results Comparison for ANSYS FLUENT

The *u*-velocity profile at the vertical centerline of the cavity and the ν-velocity profile at the horizontal centerline of the cavity are compared to Darr and Vanka results. Velocity is normalized by velocity of the moving wall.

Figure 2 Normalized *u*-Velocity at the Horizontal Centerline of the Cavity

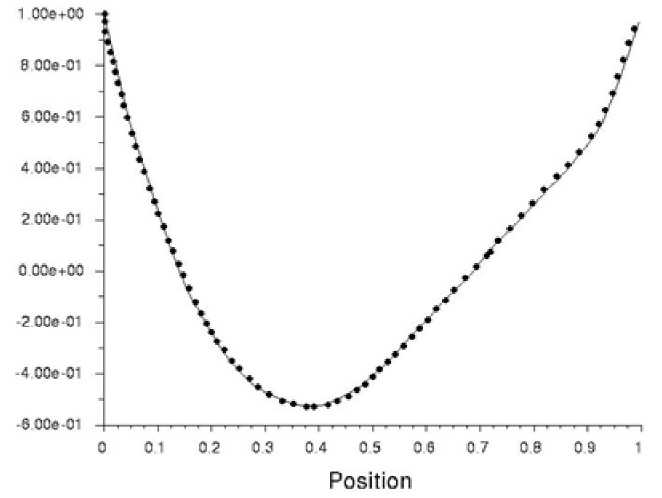
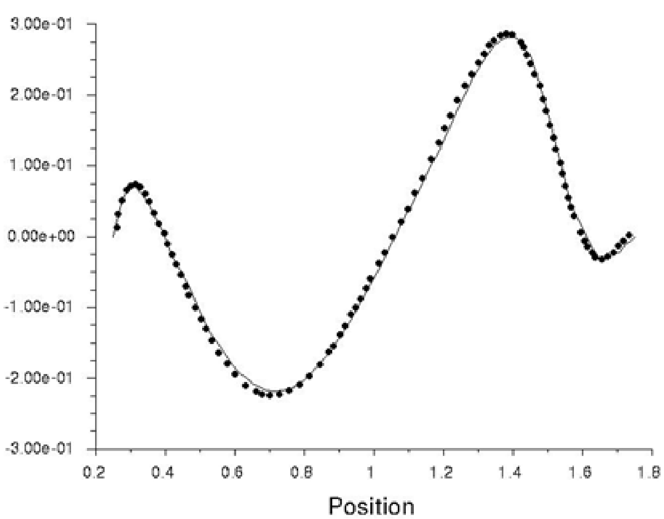


Figure 3 Normalized *v*-Velocity at the Vertical Centerline of the Cavity



Results Comparison for ANSYS CFX

The *u*-velocity profile at the vertical centerline of the cavity and the ν-velocity profile at the horizontal centerline of the cavity are compared to Darr and Vanka results. Velocity is normalized by velocity of the moving wall.

Figure 4 Normalized *u*-Velocity at the Horizontal Centerline of the Cavity

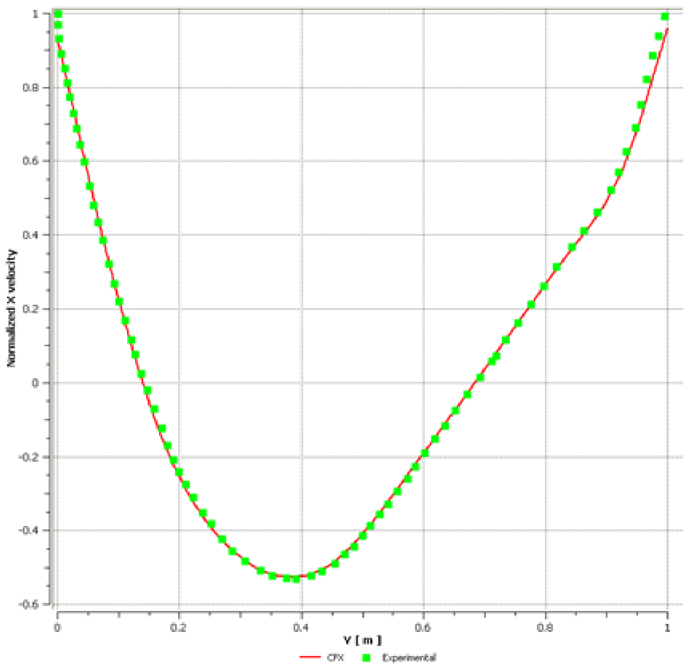
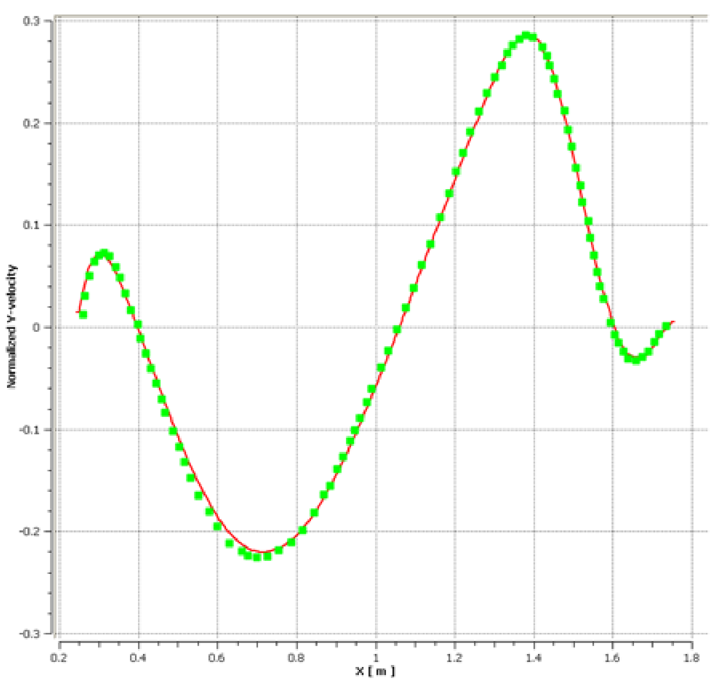


Figure 5 Normalized *v*-Velocity at the Vertical Centerline of the Cavity



VMFL055: Transitional Recirculatory Flow inside a Ventilation Enclosure

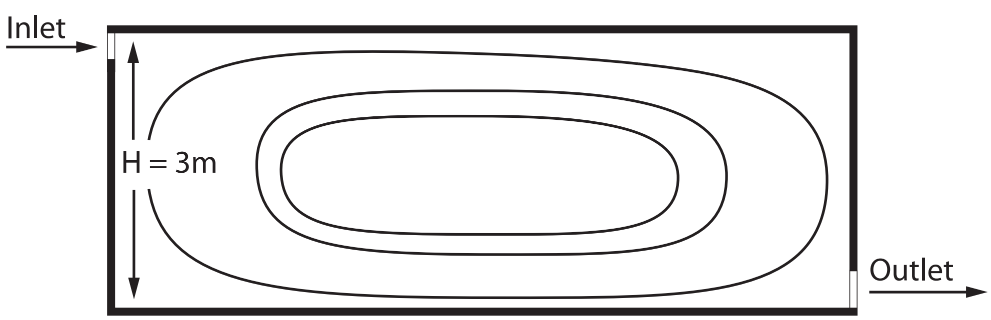
Overview

Reference	A. Restivo. *Turbulent Flow in Ventilated Rooms*. Ph.D. Thesis, University of London, U.K. 1979.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Transitional turbulence modeling (k-kl model for ANSYS FLUENT, BSL Gamma Model for ANSYS CFX)
Input File	`VMFL055_FLUENT.cas` for ANSYS FLUENT `VMFL055_CFX-BSL-Gamma.def` for ANSYS CFX

Test Case

Flow inside an enclosure similar to a ventilated room is modeled. The flow field is transitional and dominated by recirculation. Reynolds number is based on the inlet dimension and is around 5000.

Figure 1 Flow Domain



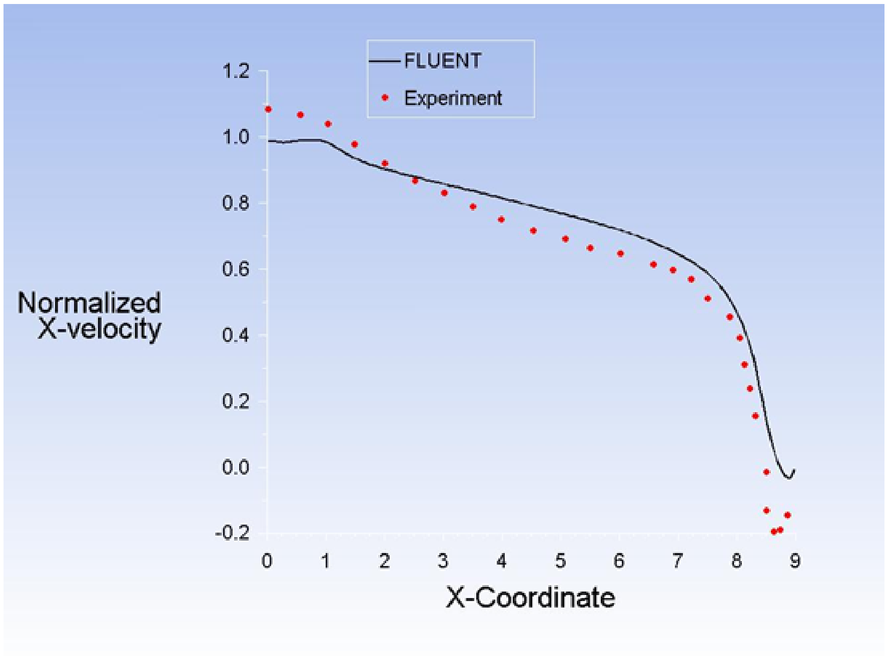
Material Properties	Geometry	Boundary Conditions
Density : 1.225 kg/m^3 Viscosity = 1.81X 10^{-5} kg/m-sec	Height of the enclosure (H) = 3 m Length of the enclosure = 9 m (3 H) Inlet : 0.056 H Outlet : 0.16 H	Inlet velocity = 0.454 m/S

Analysis Assumptions and Modeling Notes

The flow is modeled using transitional turbulence models.

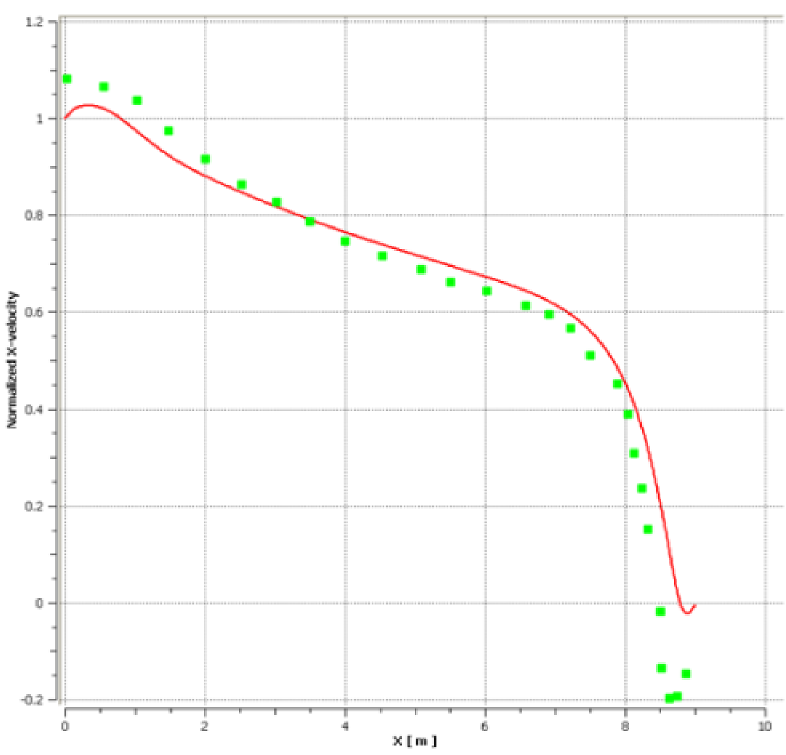
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of X-Velocity Velocity at Y = 2.916 m



Results Comparison for ANSYS CFX

Figure 3 Comparison of X-Velocity Velocity at Y = 2.916 m



VMFL056: Combined Conduction and Radiation in a Square Cavity

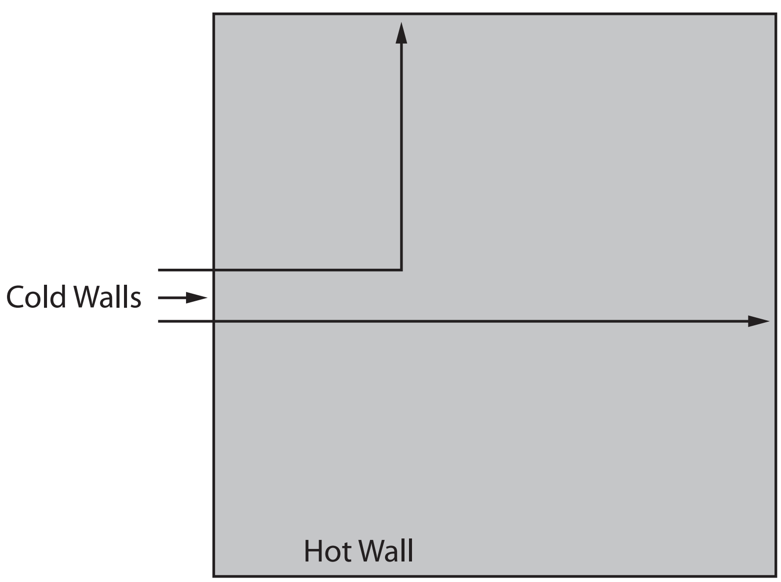
Overview

Reference	Daniel R. Rousse, Guillaume Gautier, and Jean-Francois Sacadura. "*Numerical predictions of two-dimensional conduction, convection, and radiation heat transfer. II. Validation*". *International Journal of Thermal Sciences*. Volume 39, Issue 3 (March 2000). 332-353.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Radiation modeling; discrete ordinate model in ANSYS FLUENT
Input File	`VMFL056_FLUENT.cas` for ANSYS FLUENT `VMFL056_CFX.def` for ANSYS CFX

Test Case

Coupled conduction and radiation is modeled in a square enclosure. The material properties are set to model a condition corresponding to the Conduction-Radiation parameter N = 1.0. Scattering coefficient of the medium is 0. Steady state heat transfer is modeled. One wall of the square cavity is kept at a higher temperature than the other 3 walls.

Figure 1 Flow Domain



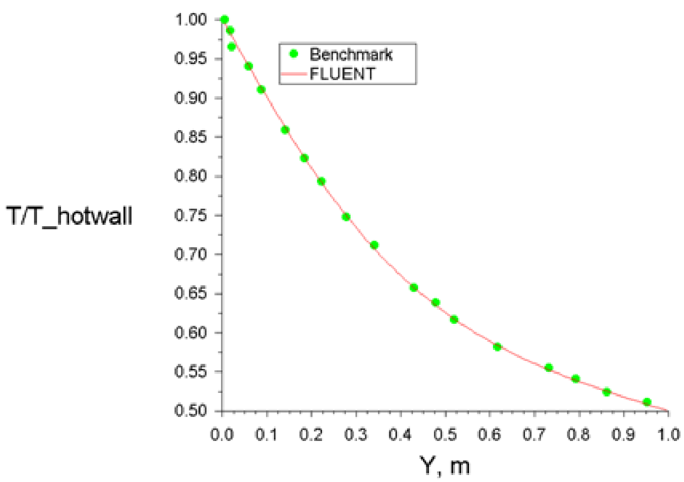
Material Properties	Geometry	Boundary Conditions
Thermal Conductivity = 1 W/m-K Absorption Coefficient = 0.228/m	Dimensions of the domain: 1 m X 1 m	Temperature of the hot wall = 100 K Temperature of the cold walls = 50 K

Analysis Assumptions and Modeling Notes

The material properties are set to model the desired conduction-radiation fraction. Radiative heat flux is only a small fraction of the total heat flux.

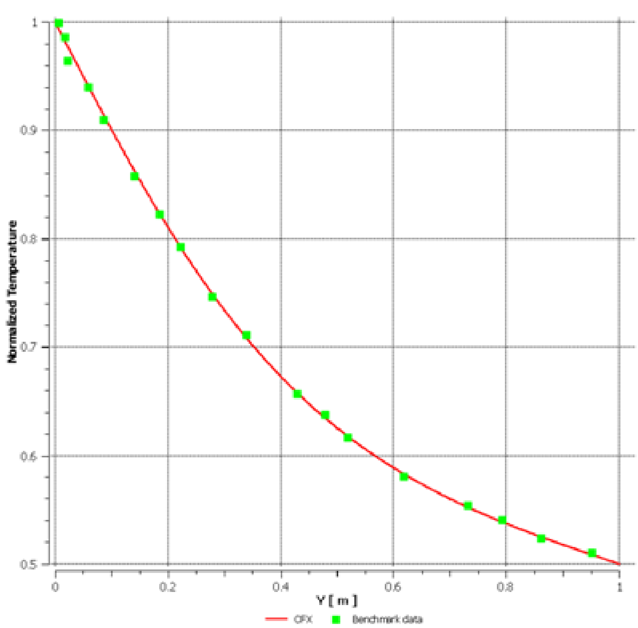
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Non-Dimensional Temperature at X = 0.5 m



Results Comparison for ANSYS CFX

Figure 3 Comparison of Non-Dimensional Temperature at X = 0.5 m



VMFL057: Radiation and Conduction in Composite Solid Layers

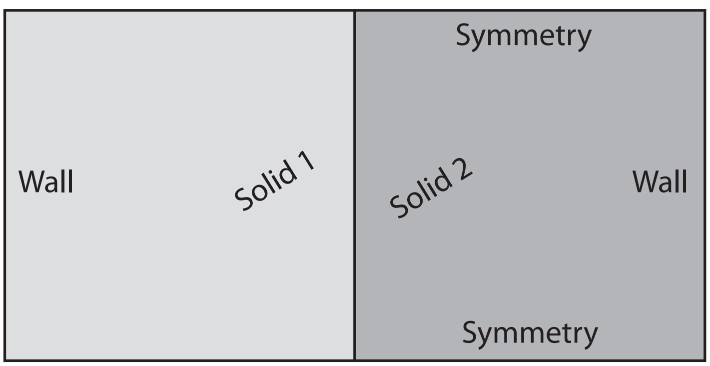
Overview

Reference	C.M. Spuckler and R. Siegel. "*Two-Flux and Diffusion Methods for Radiative Transfer in Composite Layers*". *J. Heat Transfer*, 118. 218-222. 1996.
Solver	ANSYS FLUENT
Physics/Models	Radiation modeling with DO model, participating medium with gray-band absorption
Input File	**`VMFL057_FLUENT.cas`** for ANSYS FLUENT

Test Case

Heat transfer by conduction and radiation is modeled in a composite solid domain consisting of two layers. Both the layers participate in radiation. The two layers are separated by a semi-transparent wall. The upstream and downstream sides of the domain are subjected to convective as well as radiative heat transfer.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Solid 1: Density = 2719 kg/m^3 Specific Heat = 871 J/kg-k Thermal Conductivity = 5.67 W/m-K Absorption Coefficient: gray-band Refractive Index = 1.5 Solid 2:	Dimensions of the domain: 2 m X 1 m (the two solid zones are of equal length)	Left-most wall: Convective Heat Transfer Coefficient = 56.7 W/m^2 K free stream temperature = 1000K Semi-transparent Right-most wall: Convective

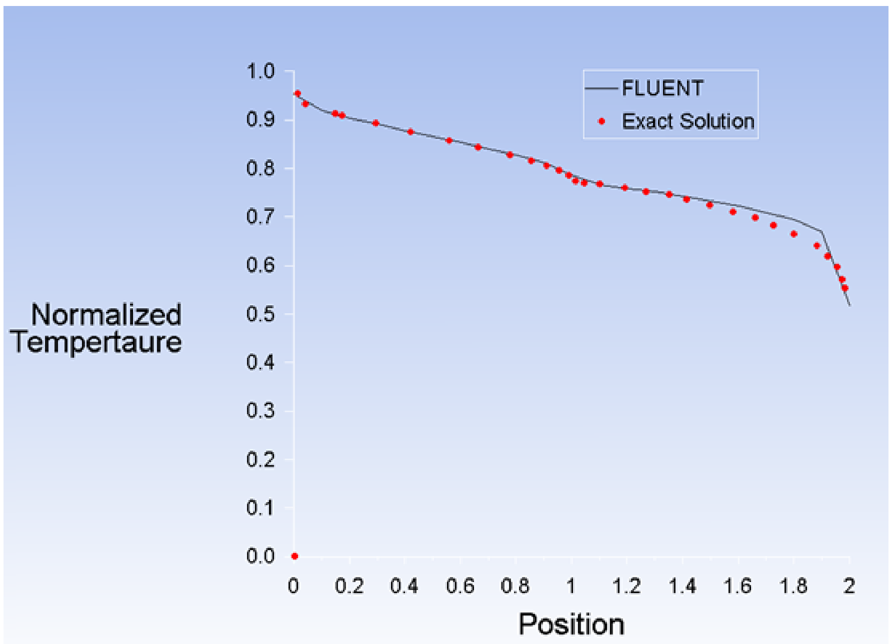
Material Properties	Geometry	Boundary Conditions
Density = 2719 kg/m^3 Specific Heat = 871 J/kg-k Thermal Conductivity = 5.67 W/m-K Absorption Coefficient: gray-band Refractive Index = 3		Heat Transfer Coefficient = 56.7 W/m^2 K free stream temperature = 250K Semi-transparent

Analysis Assumptions and Modeling Notes

Transverse boundaries of the domain are modeled as planes of symmetry.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison temperature distribution along Y = 0.5 m



VMFL058: Turbulent Flow in an Axisymmetric Diffuser

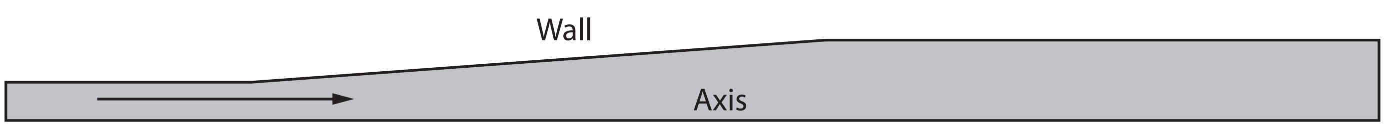
Overview

Reference	R. S. Azad and S. Z. Kassab. "*Turbulent Flow in a Conical Diffuser: Overview and implications*". Phys. Fluids A 1, 564. 1989.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Turbulent flow with adverse pressure gradient
Input File	`VMFL058_FLUENT.cas` for ANSYS FLUENT `VMFL058_CFX.def` for ANSYS CFX

Test Case

Fully developed turbulent flow is modeled in an axisymmetric diffuser. The flow is fully developed at the inlet to the diffuser.

Figure 1 Flow Domain



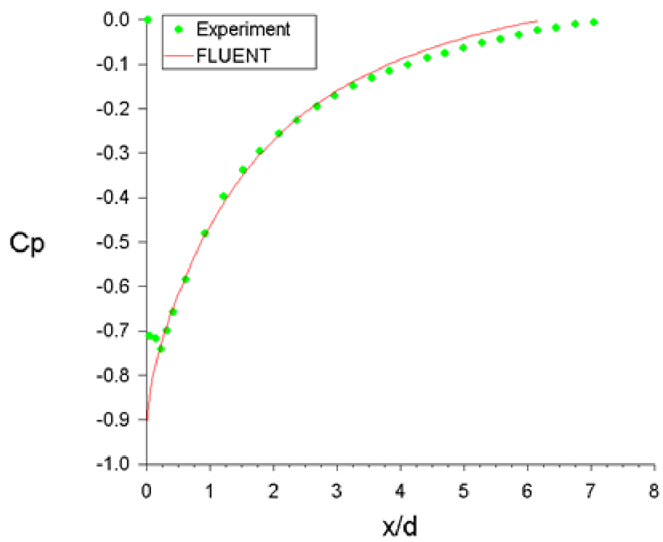
Material Properties	Geometry	Boundary Conditions
Density = 1 kg/m^3 Viscosity = 1.64 X 10^{-5} kg/m-s	Included angle for the divergent section = 8^0 Length of Inlet section (straight) = 6m Inlet radius = 1m Outlet radius = 2m	Fully developed turbulent profile at inlet with an average velocity = 1 m/s

Analysis Assumptions and Modeling Notes

Steady turbulent flow.

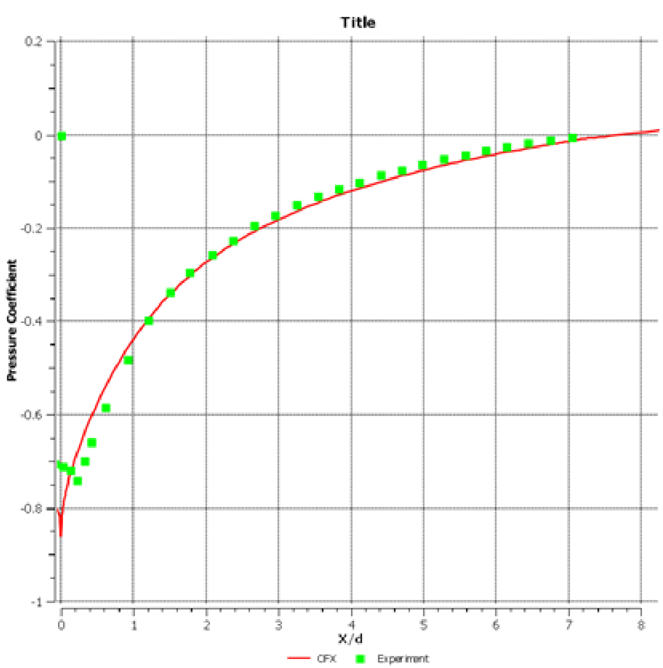
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Pressure Coefficient along the Divergent Diffuser Wall



Results Comparison for ANSYS CFX

Figure 3 Comparison of Pressure Coefficient along the Divergent Diffuser Wall



VMFL059: Conduction in a Composite Solid Block

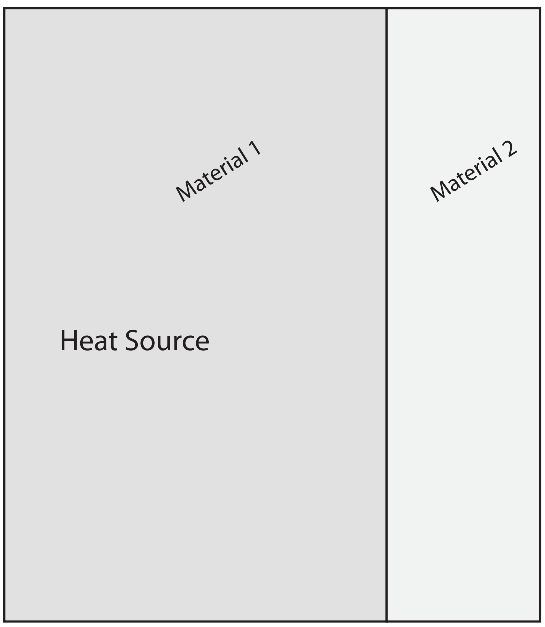
Overview

Reference	F.P. Incropera and D.P. Dewitt. "*Fundamentals of Heat and Mass Transfer*". 5th Edition. Page 117. 2006.
Solver	ANSYS FLUENT
Physics/Models	Conduction with heat source
Input File	**`VMFL059_FLUENT.cas`** for ANSYS FLUENT

Test Case

Heat conduction in a plane wall formed as composite of two materials is modeled. One of the materials has a uniform volumetric heat generation source while the other material has an outer surface exposed to convective cooling.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Material 1: Density = 2719 kg/m^3 Specific Heat = 871 J/kg-k Thermal Conductivity = 75W/m-K Material 2:	Dimensions of the domain: 0.07 m X 0.08 m Thickness of slab 1 (material 1) = 0.05 m	Left-most wall: Adiabatic Right-most wall: Convective, with Heat Transfer Coefficient = 1000 W/m^2 K and free stream temperature = 303 K Other boundaries are adiabatic walls.

Material Properties	Geometry	Boundary Conditions
Density = 8978 kg/m^3 Specific Heat = 381 J/kg-k Thermal Conductivity = 150 W/m-K		

Analysis Assumptions and Modeling Notes

Contact resistance between the slabs is neglected.

Results Comparison

Table 1 Comparison Temperatures on the Side Walls

	ANSYS FLUENT	Analytical	Ratio
Temperature of the cooled wall	378.16 K	378 K	1.0004
Temperature of the adiabatic wall on extreme left side	413.12 K	413 K	1.0003

VMFL060: Transitional Supersonic Flow over a Rearward Facing Step

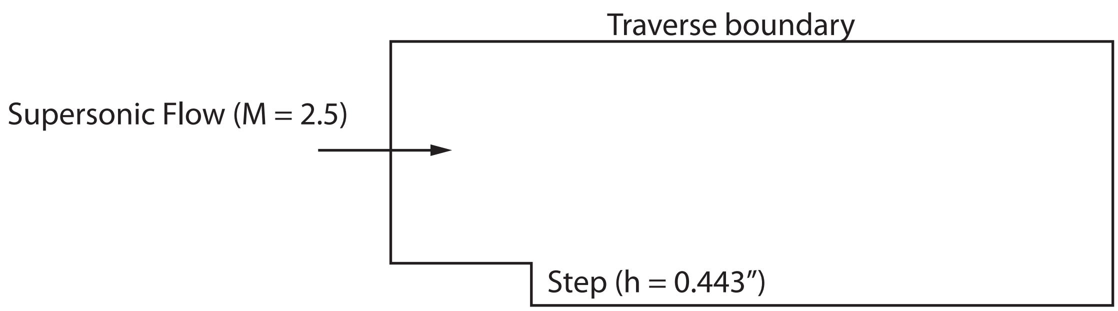
Overview

Reference	Howard E Smith. *"The Flow Field and Heat Transfer downstream of a Rearward Facing Step in Supersonic Flow".* ARL 67-0056, Aerospace Research Laboratories. Ohio, USA.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Compressible Flow, Transitional turbulence modeling (Transition SST model for ANSYS FLUENT, SST Gamma-Theta Model for ANSYS CFX)
Input File	**`VMFL060_FLUENT.cas`** for ANSYS FLUENT **`VMFL060_CFX.def`** for ANSYS CFX

Test Case

Supersonic flow with an inlet Mach number 2.5 past a backward facing step is modeled. Key features of the flow field include sudden expansion, free shear layer, recirculation zone, and oblique shock. Reynolds number of the flow (based on step height) is in the transitional range.

Figure 1 Flow Domain



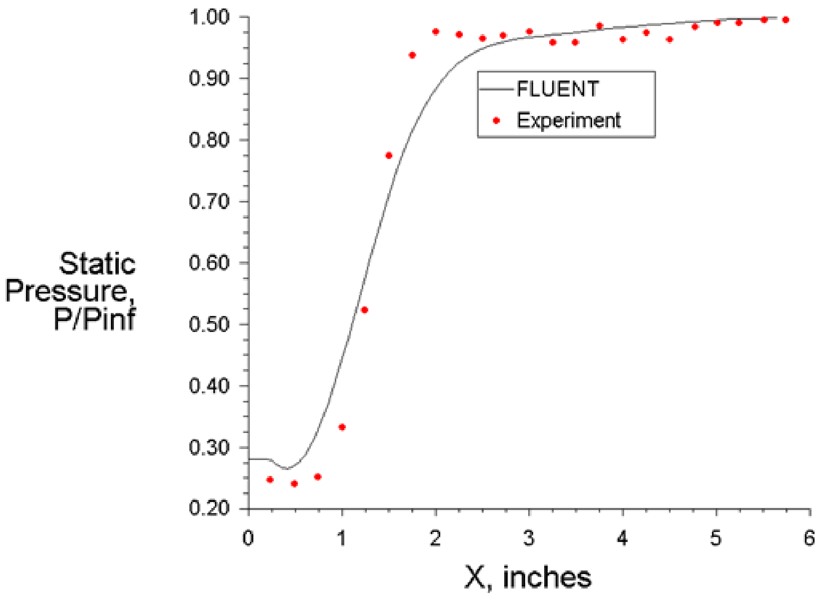
Material Properties	Geometry	Boundary Conditions
Density: Ideal gas Viscosity = 1.81X 10^{-5} kg/m-sec	Step Height = 0.443 in Inlet: 4 in upstream of the step Outlet: 12 in downstream of the step Transverse (top) boundary: 6.25 in above the step.	Total pressure at inlet = 227527 Pa Static Pressure at Inlet = 13316.6 Total temperature at inlet = 344.44 K Transverse boundary modeled as far-field (ANSYS FLUENT)/ Supersonic outlet (ANSYS CFX) Walls are modeled as adiabatic

Analysis Assumptions and Modeling Notes

The flow is modeled using transitional turbulence models.

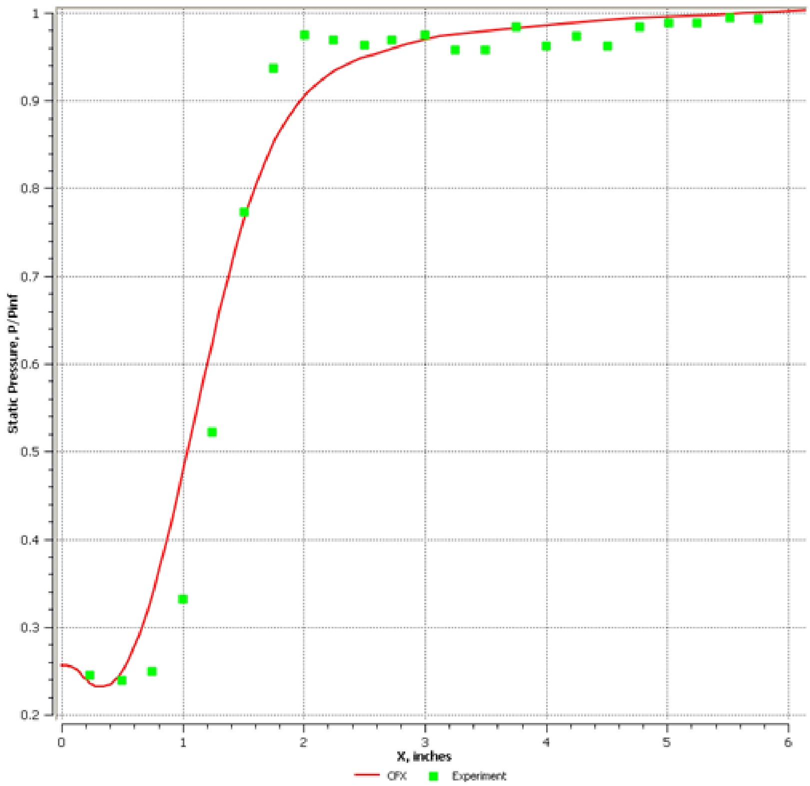
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Non-Dimensionalized Static Pressure along the Stepped Wall Downstream of the Corner



Results Comparison for ANSYS CFX

Figure 3 Comparison of Non-Dimensionalized Static Pressure along the Stepped Wall Downstream of the Corner



VMFL061: Surface to Surface Radiative Heat Transfer between Two Concentric Cylinders

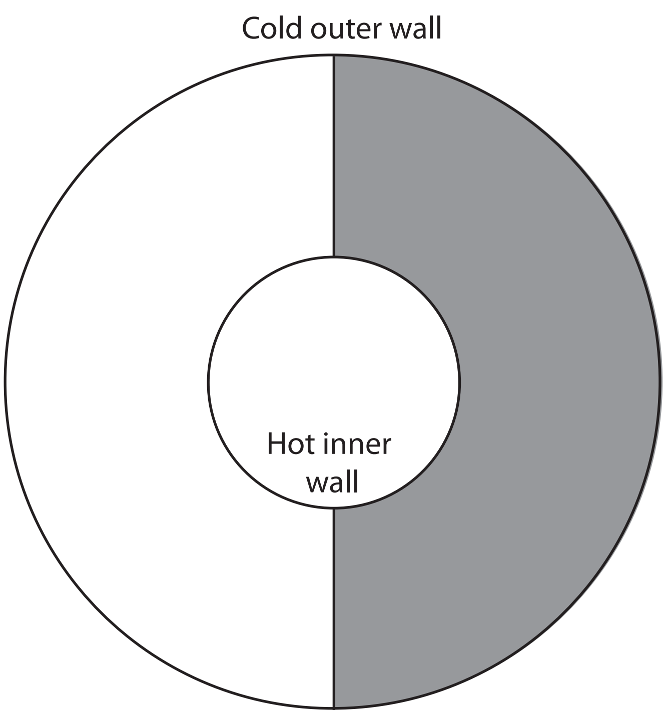
Overview

Reference	F.P. Incropera and D.P. Dewitt. "*Fundamentals of Heat and Mass Transfer*". 4th Edition. New York City, New York: John Wiley & Sons, Inc. 1996.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Radiation Modeling (S2S Model in ANSYS FLUENT, Monte Carlo Method in ANSYS CFX)
Input File	**`VMFL061_FLUENT.cas`** for ANSYS FLUENT **`VMFL061_CFX.def`** for ANSYS CFX

Test Case

Radiative heat transfer between two cylindrical surfaces forming a concentric annulus is modeled. There is no participating medium. Due to symmetry only the shaded portion of the domain in *Figure 1* (p. 209) is modeled.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Material does not participate in the energy transfer.	Radius of inner wall = 0.04625 m	Temperature of inner wall = 700 K

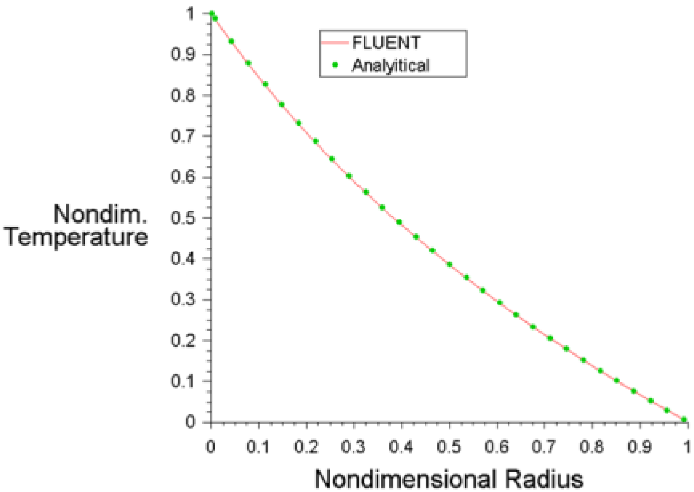
Material Properties	Geometry	Boundary Conditions
	Radius of outer wall = 0.0178 m	Temperature of outer wall = 300 K

Analysis Assumptions and Modeling Notes

Because there is no flow of mass, only the energy equation is solved. Radiation models are used for the simulation. Heat transfer is purely due to radiation between the two surfaces.

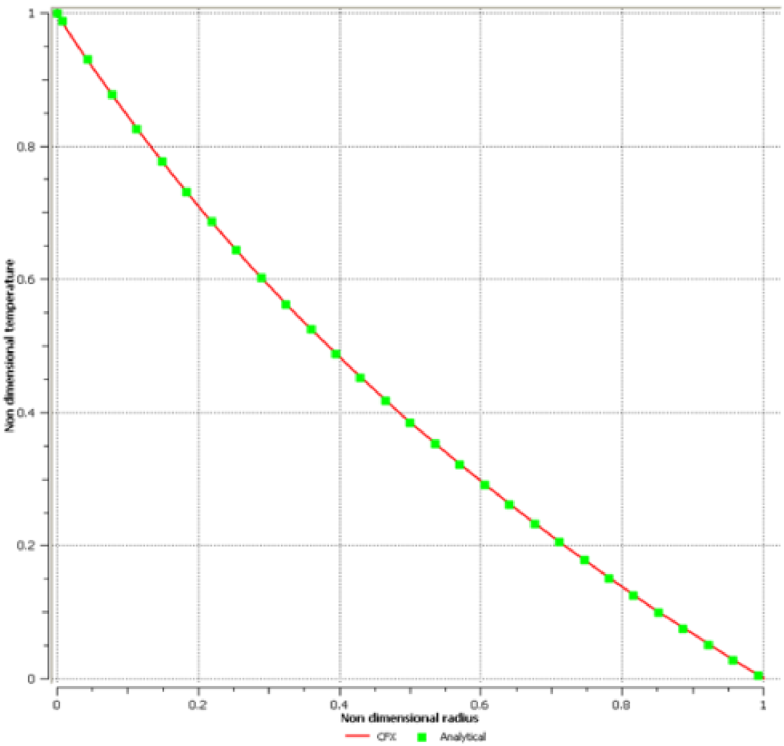
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Temperature Variation along Radius



Results Comparison for ANSYS CFX

Figure 3 Comparison of Temperature Variation along Radius



VMFL062: Fully Developed Turbulent Flow Over a "Hill"

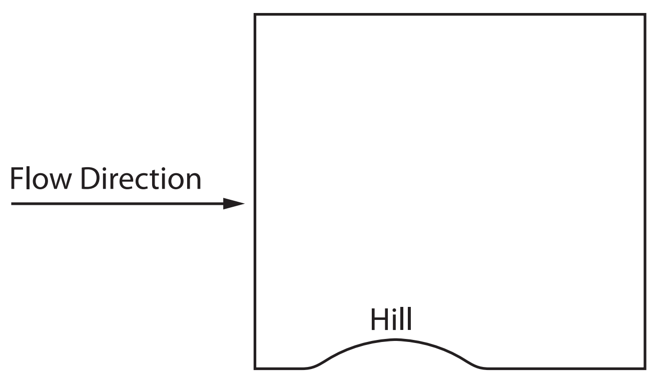
Overview

Reference	V. Baskaran, A. J. Smits, and P. N. Joubert. "*A turbulent flow over a curved hill Part 1. Growth of an internal boundary layer*". *Journal of Fluid Mechanics*, (1987), 182: 47-83.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Low Re k-e Model in ANSYS FLUENT, k-e Model in ANSYS CFX
Input File	**`VMFL062_FLUENT.cas`** for ANSYS FLUENT **`VMFL062_CFX.def`** for ANSYS CFX

Test Case

Flow over a "hill" geometry with separation and reattachment is modeled. Fully developed turbulent profile is specified at the inlet.

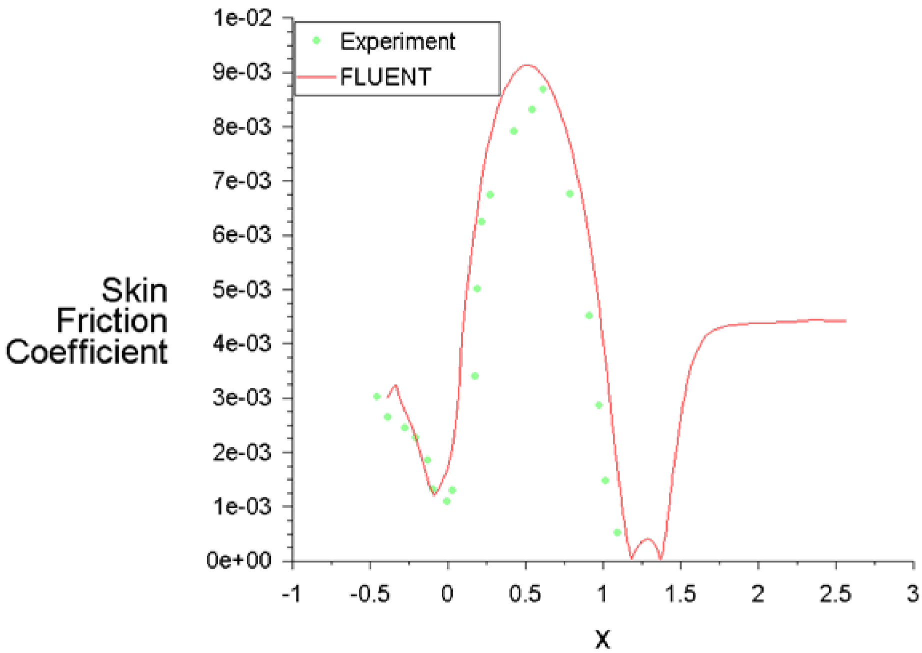
Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density : 1 kg/m^3 Viscosity = 7.5188e-07 kg/m-s	Height of the hill = 20.5 mm	Fully developed profiles are specified at the inlet for (i) Velocity, (ii) Turbulent kinetic Energy, and (iii) Eddy dissipation rate. Average velocity at inlet = 1 m/s

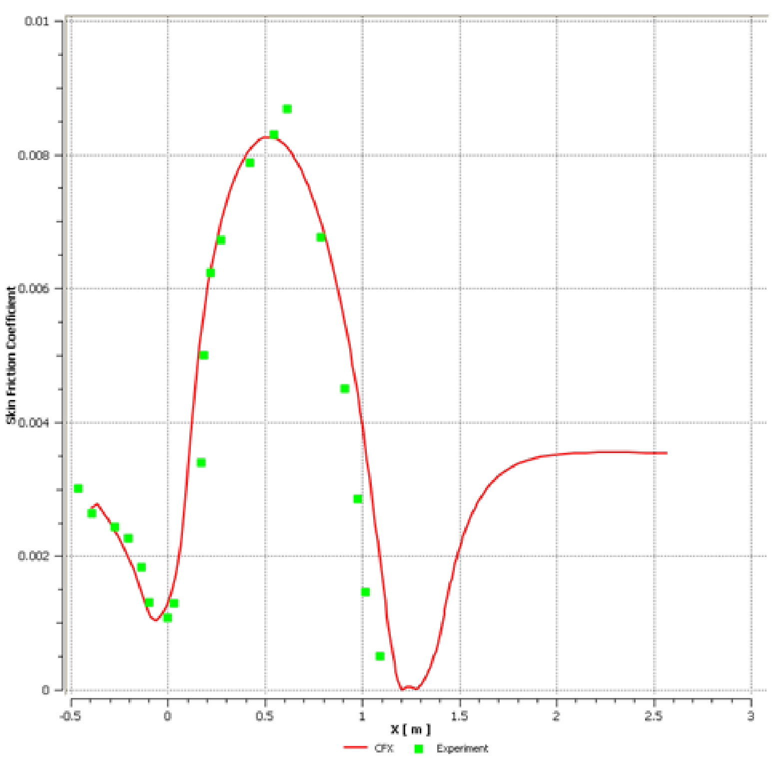
Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Skin Friction along the Wall



Results Comparison for ANSYS CFX

Figure 3 Comparison of Skin Friction along the Wall



VMFL063: Separated Laminar Flow over a Blunt Plate

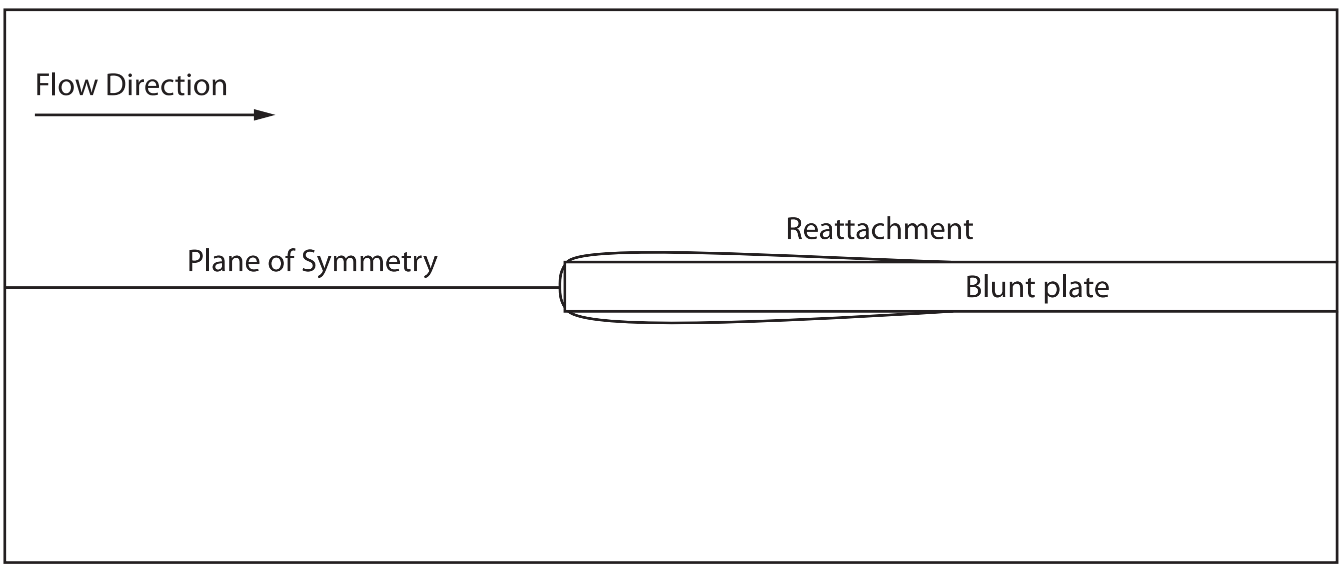
Overview

Reference	J.C . Lane and R.I. Loehrke. "*Leading Edge Separation from a Blunt Plate at Low Reynolds Number*". *Transactions of ASME*, Volume 102 (December 1980): 494-496.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Laminar flow, high resolution numerical models
Input File	`VMFL063_FLUENT.cas` for ANSYS FLUENT `VMFL063_CFX.def` for ANSYS CFX

Test Case

The flow separation over a blunt leading edge in laminar flow is modeled. The flow separates and reattaches along the plate. The reattachment length predicted by the solvers is validated against experimental results. Due to symmetry, only half of the domain shown in *Figure 1* (p. 215) is modeled. The Reynolds number based on plate thickness is 227.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 1.7894 X 10^{-5} kg/m-s	Thickness of the plate, 2t = 90 mm Length of the plate = 1500 mm	Velocity at inlet = 0.045133 m/s

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Reattachment Length

	ANSYS FLUENT	Experiment	Ratio
Non-dimensionalized Reattachment length (L_R/2t)	3.8	4.0	0.95

Results Comparison for ANSYS CFX

Table 2 Comparison of Reattachment Length

	ANSYS CFX	Experiment	Ratio
Non-dimensionalized Reattachment length (L_R/2t)	4.37	4.0	1.093

VMFL064: Low Reynolds Number Flow in a Channel with Sudden Asymmetric Expansion

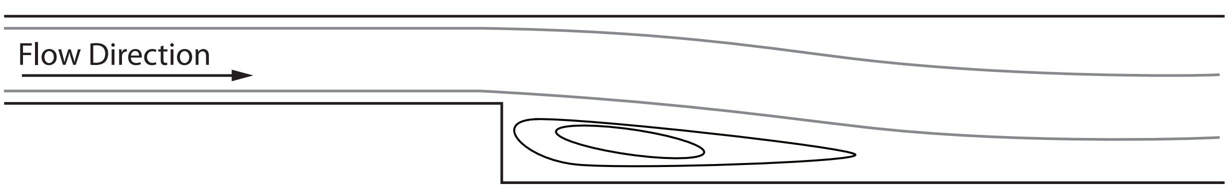
Overview

Reference	1. B. Armaly, F. Durst, J. Pereira, and B. Schönung. "*Experimental and theoretical investigation of a backward-facing step*". *J. Fluid Mech.* 127 (1983): 473. 2. C.J. Freitas. "*Perspective: Selected Benchmarks from Commercial CFD Codes*". *J. Fluids Eng*. Volume 117, Issue 2 (June 1995): 208.
Solver	ANSYS FLUENT, ANSYS CFX
Physics/Models	Laminar flow, Separation, and Reattachment
Input File	`VMFL064_FLUENT.cas` for ANSYS FLUENT `VMFL064_CFX.def` for ANSYS CFX

Test Case

Laminar flow in a channel with a backward facing step expansion is modeled. The channels section upstream of the expansion is long enough to ensure fully developed laminar profile. The reattachment length predicted by the solvers is validated against experimental results. Reynolds number based on D (equal to twice the channel height at inlet) is 200. The domain extends to about 40 times the step-height upstream and over 20 times the step-height downstream.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 1.5 X 10^{-5} kg/m-s	Step height, s = 4.9 mm Channel height at inlet = 5.2 mm Length of inlet section = 200 mm Length downstream of the step = 100 mm	Velocity at inlet = 0.288462 m/s No-slip condition at the walls

Analysis Assumptions and Modeling Notes

The flow is fully developed before the step. Reattachment length is measured from the reversal of the sign of the wall shear along the flow direction.

Results Comparison for ANSYS FLUENT

Table 1 Comparison of Reattachment Length

	ANSYS FLUENT	Experiment	Ratio
Non-dimensionalized Reattachment length (L_R/Step-height)	4.93	5.0	0.986

Results Comparison for ANSYS CFX

Table 2 Comparison of Reattachment Length

	ANSYS CFX	Experiment	Ratio
Non-dimensionalized Reattachment length (L_R/Step-height)	4.90	5.0	0.98

VMFL065: Swirling Turbulent Flow Inside a Diffuser

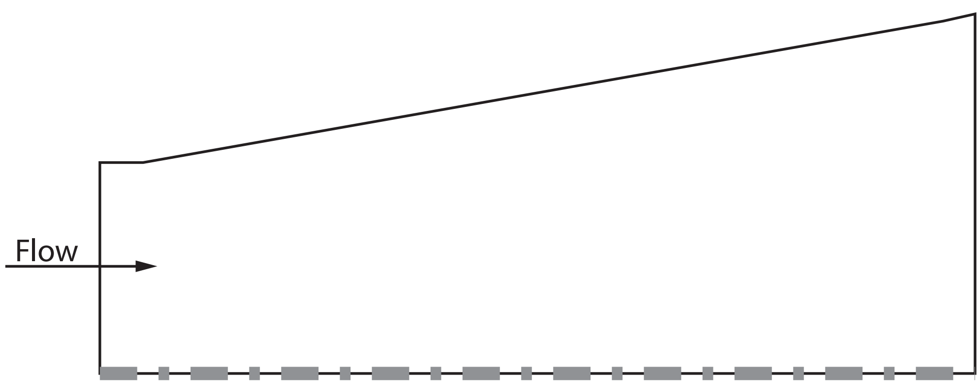
Overview

Reference	P.D. Clausen, S.G. Koh, and D.H. Wood. "*Measurements of a Swirling Turbulent Boundary Layer Developing in a Conical Diffuser.*" *Experimental Thermal and Fluid Science*. Volume 6, Issue 1 (January 1993): 39-48.
Solver	ANSYS FLUENT
Physics/Models	Turbulent flow, swirl velocity, Reynolds stress model for turbulence
Input File	`VMFL065_FLUENT.cas` for ANSYS FLUENT

Test Case

Turbulent flow with a strong swirl component is modeled in an axisysmmetric diffuser. The swirl component of the velocity has a dominant effect on the flow field inside the diffuser.

Figure 1 Flow Domain



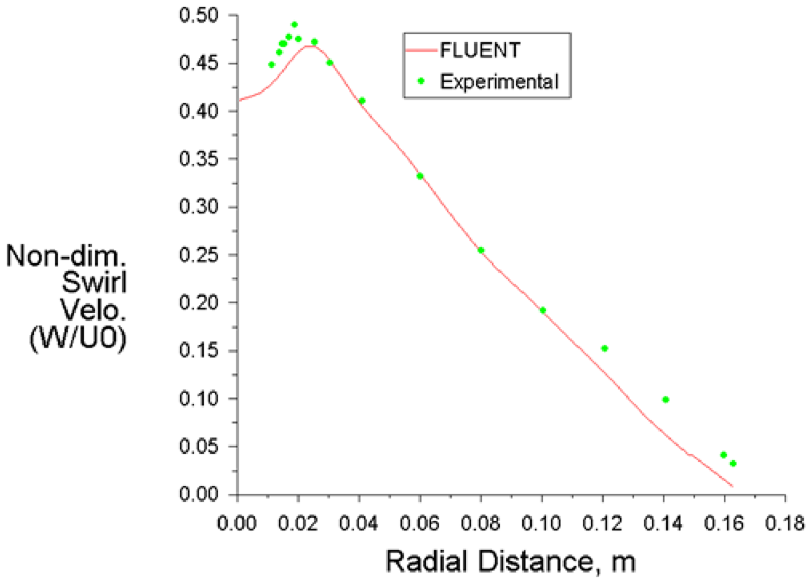
Material Properties	Geometry	Boundary Conditions
Density: 1 kg/m^3 Viscosity: 1.293 X 10^{-6} kg/m-s	Length of the straight inlet section = 25 mm Length of the diffuser (divergent section) = 510 mm Inlet Diameter = 260 mm Outlet Diameter = 440 mm	Fully developed turbulent profile for velocity, k and ε at inlet (with average axial inlet velocity = 1 m/s) No-slip condition at the walls

Analysis Assumptions and Modeling Notes

RS model is used for turbulence due to the strong swirl component.

Results Comparison for ANSYS FLUENT

Figure 2 Comparison of Swirl Velocity at X = 0.175 m



VMFL066: Radiative Heat Transfer in a Rectangular Enclosure with Participating Medium

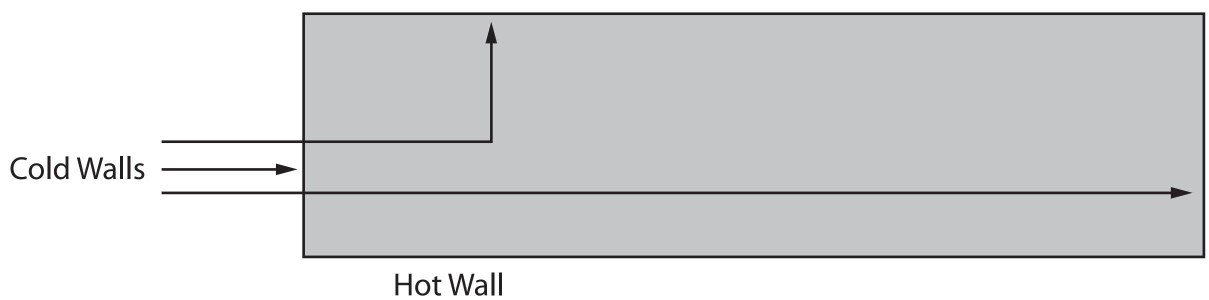
Overview

Reference	G.D Raithby and E.H. Chui. "*A Finite Volume Method for Predicting a Radiant Heat Transfer in Enclosoures with Participating Media*". *Journal of Heat Transfer*. Volume 112 (May 1990): 415-423.
Solver	ANSYS FLUENT
Physics/Models	Radiation modeling, discrete Ordinate Model in ANSYS FLUENT
Input File	`VMFL066_FLUENT.cas` for ANSYS FLUENT

Test Case

Two dimensional radiative heat transfer in an enclosure with one hot wall and three cold walls at equal temperature is modeled. The enclosure is a rectangular cavity with a length-to-width ratio of 5. For the problem being considered, $\sigma^s L_y = 1.0$, where σ^s is the scattering coefficient and L_y is the normal distance between the hot wall and the cold wall opposite to it.

Figure 1 Flow Domain



Material Properties	Geometry	Boundary Conditions
Scattering coefficient = 0.5/m	Dimensions of the domain: 10 m x 2 m	Temperature of the hot wall = 200 K Temperature of he cold walls = 100 K

Analysis Assumptions and Modeling Notes

Isotropic scattering and radiative equilibrium are assumed.

Results Comparison for ANSYS FLUENT

Figure 2

Comparison of Non-Dimensional Heat Flux along the Hot Wall