Chapter 6: Modeling Transient Compressible Flow

This tutorial is divided into the following sections:

- 6.1. Introduction
- 6.2. Prerequisites
- 6.3. Problem Description
- 6.4. Setup and Solution
- 6.5. Summary
- 6.6. Further Improvements

6.1. Introduction

In this tutorial, ANSYS Fluent's density-based implicit solver is used to predict the time-dependent flow through a two-dimensional nozzle. As an initial condition for the transient problem, a steady-state solution is generated to provide the initial values for the mass flow rate at the nozzle exit.

This tutorial demonstrates how to do the following:

- Calculate a steady-state solution (using the density-based implicit solver) as an initial condition for a transient flow prediction.
- Define a transient boundary condition using a user-defined function (UDF).
- Use dynamic mesh adaption for both steady-state and transient flows.
- Calculate a transient solution using the second-order implicit transient formulation and the density-based implicit solver.
- Create an animation of the transient flow using ANSYS Fluent's transient solution animation feature.

6.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

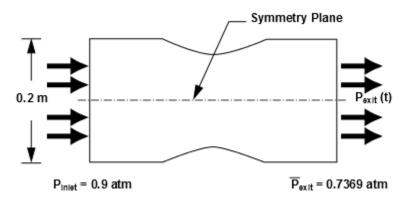
- Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121)

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

6.3. Problem Description

The geometry to be considered in this tutorial is shown in Figure 6.1: Problem Specification (p. 268). Flow through a simple nozzle is simulated as a 2D planar model. The nozzle has an inlet height of 0.2 m, and the nozzle contours have a sinusoidal shape that produces a 20% reduction in flow area. Symmetry allows only half of the nozzle to be modeled.

Figure 6.1: Problem Specification



6.4. Setup and Solution

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

- 6.4.1. Preparation
- 6.4.2. Reading and Checking the Mesh
- 6.4.3. Solver and Analysis Type
- 6.4.4. Models
- 6.4.5. Materials
- 6.4.6. Operating Conditions
- 6.4.7. Boundary Conditions
- 6.4.8. Solution: Steady Flow

6.4.9. Enabling Time Dependence and Setting Transient Conditions

- 6.4.10. Specifying Solution Parameters for Transient Flow and Solving
- 6.4.11. Saving and Postprocessing Time-Dependent Data Sets

6.4.1. Preparation

To prepare for running this tutorial:

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

Note

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

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.

- a. Click ANSYS Fluent under Product.
- b. Click 18.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click the **unsteady_compressible_R180.zip** link to download the input files.
- 7. Unzip the unsteady_compressible_R180.zip file you downloaded to your working folder.

The files nozzle.msh and pexit.c can be found in the unsteady_compressible folder created after unzipping the file.

8. Use Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

For more information about the Fluent Launcher, see starting ANSYS Fluent using the Fluent Launcher in the Fluent Getting Started Guide.

- 9. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.
- 10. Ensure that **Serial** is selected under **Processing Options**.
- 11. Disable the **Double Precision** option.

6.4.2. Reading and Checking the Mesh

1. Read the mesh file nozzle.msh.



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

2. Check the mesh.



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

3. Verify that the mesh size is correct.



💶 Scale N	/lesh			
Domain Ext	tents			Scaling
Xmin (m)	-0.1	Xmax (m)		 Convert Units Specify Scaling Factors
Ymin (m)	0	Ymax (m)	0.1	Mesh Was Created In
View Lengt	h Unit In T			Scaling Factors X 1 Y 1 Scale Unscale
		C	lose Help	

Close the Scale Mesh dialog box.

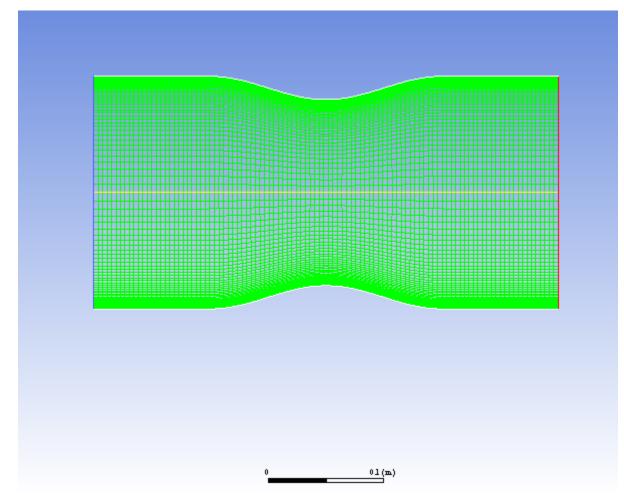
4. Mirror the mesh across the centerline (Figure 6.2: 2D Nozzle Mesh Display with Mirroring (p. 271)).

$\blacksquare Viewing \rightarrow Display \rightarrow Views...$

E Views		
Views	Actions	Mirror Planes [1/1]
back front	Default Auto Scale Previous	symmetry
	Save Delete	
	Read Write	Define Plane Periodic Repeats
Save Name view-0	- vince	Define
Apply C	Camera Clo	se Help

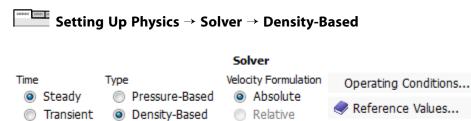
- a. Select symmetry in the Mirror Planes selection list.
- b. Click **Apply** to refresh the display.
- c. Close the **Views** dialog box.





6.4.3. Solver and Analysis Type

1. Select the solver settings.



a. In the Solver group of the Setting Up Physics tab, select Density-Based from the Type list.

The density-based implicit solver is the solver of choice for compressible, transonic flows without significant regions of low-speed flow. In cases with significant low-speed flow regions, the pressure-based solver is preferred. Also, for transient cases with traveling shocks, the density-based explicit solver with explicit time stepping may be the most efficient.

Planar

b. Retain the default selection of Steady from the Time list.

Note

You will solve for the steady flow through the nozzle initially. In later steps, you will use these initial results as a starting point for a transient calculation.

2. For convenience, change the unit of measurement for pressure.

The pressure for this problem is specified in atm, which is not the default unit in ANSYS Fluent. You must redefine the pressure unit as atm.

100001 10000 10	Setting	Up	Domain	\rightarrow	$Mesh \rightarrow$	Units
-----------------	---------	----	--------	---------------	--------------------	-------

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

a. Select pressure in the Quantities selection list.

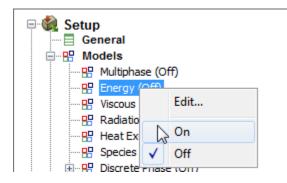
Scroll down the list to find **pressure**.

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

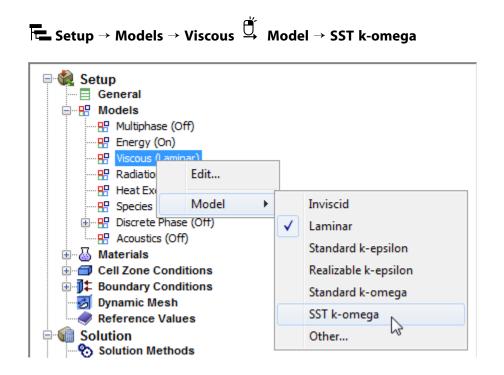
6.4.4. Models

1. Enable the energy equation.





2. Select the k-omega SST turbulence model.



6.4.5. Materials

1. Define the settings for air, the default fluid material.



ame	Material Type		Order Materials by
air	fluid	+	Name
hemical Formula	FLUENT Fluid Materials		Chemical Formula
	ar	•	FLUENT Database
	Modure		User-Defined Database
	none		
roperties		- C	
Density (kg/m3)	deal-gas 🔸 Edt	-	
Cp (Specific Heat) ()/kg-k)	constant 🔹 Edit		
	1006.43	8	
Thermal Conductivity (w/m-k)			
memai caracemia (whing	constant 🔹 Edit		
	0.0242		
Viscosity (kg/m-s)	constant • Edit	1.00	
	1.7894e-05		

a. Select **ideal-gas** from the **Density** drop-down list in the **Properties** group box, so that the ideal gas law is used to calculate density.

Note

ANSYS Fluent automatically enables the solution of the energy equation when the ideal gas law is used, in case you did not already enable it manually in the **Energy** dialog box.

- b. Retain the default values for all other properties.
- c. Click the **Change/Create** button to save your change.
- d. Close the Create/Edit Materials dialog box.

6.4.6. Operating Conditions

1. Define the operating pressure.

Setting Up Physics \rightarrow Solver \rightarrow Operating Conditions...

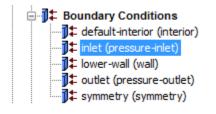
Operating Conditions	-
Pressure	Gravity
Operating Pressure (atm) 0	Gravity
Reference Pressure Location	
X (m) 0	
Y (m) 0	
Z (m) 0	
OK Cancel Help	,

- a. Enter 0 atm for **Operating Pressure**.
- b. Click **OK** to close the **Operating Conditions** dialog box.

Since you have set the operating pressure to zero, you will specify the boundary condition inputs for pressure in terms of absolute pressures when you define them in the next step. Boundary condition inputs for pressure should always be relative to the value used for operating pressure.

6.4.7. Boundary Conditions

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





	Pressure Inle	et							×
	Zone Name								
	inlet								
	Momentum Thermal Radiation Species DPM Multiphase Potential UDS								
	Reference Frame Absolute								
	Gauge Total Pressure (atm) 0.9 constant								
	Supersonic/Initial Gauge Pressure (atm) 0.7369 constant								
	Direction Specification Method Normal to Boundary								
	Turbulence								
	Specification Method Intensity and Viscosity Ratio								•
Turbulent Intensity (%) 1.5							P		
	Turbulent Viscosity Ratio 10								
			OK	Cancel	Help				

- a. Enter 0.9 atm for Gauge Total Pressure.
- b. Enter 0.7369 atm for Supersonic/Initial Gauge Pressure.

The inlet static pressure estimate is the mean pressure at the nozzle exit. This value will be used during the solution initialization phase to provide a guess for the nozzle velocity.

- c. Retain **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.
- d. Enter 1.5% for **Turbulent Intensity**.
- e. Retain the setting of 10 for **Turbulent Viscosity Ratio**.
- f. Click **OK** to close the **Pressure Inlet** dialog box.
- 2. Define the boundary conditions for the nozzle exit (outlet).

E Setup \rightarrow Boundary Conditions \rightarrow outlet (pressure-outlet) $\stackrel{\frown}{\sqcup}$ Edit...

nermal Radiation	Species	DPM Multiphas	e Potential	UDS
low Reference Frame	Absolute			*
Gauge Pressure (atm) 0.7369		constant	•
Specification Method	Normal to Bou	indary		•
re Specification	2			
w Rate				
Turbulence				
Specification Method	Intensity and \	iscosity Ratio		•
Back	flow Turbulent	Intensity (%) 1.5		P
Backf	low Turbulent	Viscosity Ratio 10		P
odel I				
	ow Reference Frame Gauge Pressure (Specification Method e Specification v Rate Turbulence pecification Method Back Backf	ow Reference Frame Absolute Gauge Pressure (atm) 0.7369 Specification Method Normal to Bou e Specification v Rate Turbulence pecification Method Intensity and V Backflow Turbulent Backflow Turbulent	ow Reference Frame Absolute Gauge Pressure (atm) 0.7369 Specification Method Normal to Boundary e Specification v Rate Turbulence pecification Method Intensity and Viscosity Ratio Backflow Turbulent Intensity (%) 1.5 Backflow Turbulent Viscosity Ratio 10	ow Reference Frame Absolute Gauge Pressure (atm) 0.7369 constant Specification Method Normal to Boundary e Specification v Rate Turbulence pecification Method Intensity and Viscosity Ratio Backflow Turbulent Intensity (%) 1.5 Backflow Turbulent Viscosity Ratio 10

- a. Enter 0.7369 atm for Gauge Pressure.
- b. Retain **Intensity and Viscosity Ratio** from the **Specification Method** drop-down list in the **Turbulence** group box.
- c. Enter 1.5% for **Backflow Turbulent Intensity**.
- d. Retain the setting of 10 for Backflow Turbulent Viscosity Ratio.

If substantial backflow occurs at the outlet, you may need to adjust the backflow values to levels close to the actual exit conditions.

e. Click OK to close the Pressure Outlet dialog box.

6.4.8. Solution: Steady Flow

In this step, you will generate a steady-state flow solution that will be used as an initial condition for the time-dependent solution.

1. Define the solution parameters.



Solution Methods

Implicit 👻	
Flux Type	
Roe-FDS 👻	
Spatial Discretization	_
Gradient	1
Least Squares Cell Based 👻	
Flow	
Second Order Upwind 🗸	
Turbulent Kinetic Energy	
Second Order Upwind 🗸	
Specific Dissipation Rate	
Second Order Upwind 🗸	
Transient Formulation	-
▼	
Non-Iterative Time Advancement Frozen Flux Formulation Pseudo Transient High Order Term Relaxation Options	
Convergence Acceleration For Stretched Meshes	
Default	

- a. Retain the default selection of **Least Squares Cell Based** from the **Gradient** drop-down list in the **Spatial Discretization** group box.
- b. Select Second Order Upwind from the Turbulent Kinetic Energy and Specific Dissipation Rate dropdown lists.

Second-order discretization provides optimum accuracy.

2. Modify the Courant Number.

Solving \rightarrow Controls \rightarrow Controls...

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

a. Enter 50 for the Courant Number.

Note

The default Courant number for the density-based implicit formulation is 5. For relatively simple problems, setting the Courant number to 10, 20, 100, or even higher value may be suitable and produce fast and stable convergence. However, if you encounter convergence difficulties at the startup of the simulation of a properly set up problem, then you should consider setting the Courant number to its default value of 5. As the solution progresses, you can start to gradually increase the Courant number until the final convergence is reached.

- b. Retain the default values for the Under-Relaxation Factors.
- 3. Enable the plotting of residuals.



Residual Monitors			—		
Options Print to Console Plot Window 1 Curves Axes Iterations to Plot 1000	Equations Residual Monitor continuity Image: Continuity x-velocity Image: Continuity y-velocity Image: Continuity energy Image: Continuity				
Iterations to Store	Residual Values Normalize Scale Compute Local Scale	Iterations 5	Convergence Criterion		
OK Plot Renormalize Cancel Help					

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Select none from the Convergence Criterion drop-down list.
- c. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Create the surface report definition for mass flow rate at the flow exit.

Solving \rightarrow Reports \rightarrow Definitions \rightarrow New \rightarrow Surface Report \rightarrow Mass Flow Rate...

Name	Report Type	
mass_flowrate_out	Mass Flow Rate	ં
Options Per Surface Average Over 1 Report Files [0/0]	Custom Vectors Vectors of Custom Vectors Field Variable Pressure Static Pressure Surfaces Filter Text	
Report Plots [0/0]	default-interior	
Create Create Report File Report Plot Frequency 1 Print to Console		
Create Output Parameter	New Surface 🔻	
	OK Compute Cancel Help	

- a. Enter mass_flowrate_out for Name.
- b. Select **outlet** in the **Surfaces** selection list.
- c. In the Create group box, enable Report File, Report Plot and Print to Console.

Note

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

d. Click OK to close the Surface Report Definition dialog box.

mass_flowrate_out-rplot and mass_flowrate_out-rfile are automatically generated by Fluent and appear in the tree (under Solution/Monitors/Report Plots and Solution/Monitors/Report Files, respectively).

e. Modify the output file name.

lame	
mass_flowrate_out-rfile	
Available Report Definitions [0/0]	Selected Report Definitions [0/1]
	mass_flowrate_out
	Add>>
	< <remove]<="" td=""></remove>
Output File Base Name	New V Edt
	New V Edt
noz_ss.out Bro	
noz_ss.out Bro Full File Name Get Data Every	
noz_ss.out Brow Full File Name Get Data Every 1 💿 [iteration	wse
noz_ss.out Bro Full File Name Get Data Every	wse

Solution \rightarrow Monitors \rightarrow Report Files \rightarrow mass_flowrate_out-rfile $\stackrel{\bigcirc}{\longrightarrow}$ Edit...

- i. Enter **noz_ss.out** for **Output File Base Name**.
- ii. Click **OK** to close the **Edit Report File** dialog box.
- 5. Save the case file (noz_ss.cas.gz).

File
$$\rightarrow$$
 Write \rightarrow Case...

6. Initialize the solution.

Solving \rightarrow Initialization				
Initialization				
		Patch		
Hybrid More Settings Reset Statistics				
Standard	Options	Reset DPM	t = 0 Initializ	

- a. Keep the **Method** at the default of **Hybrid**.
- b. Click Initialize.
- 7. Set up gradient adaption for dynamic mesh refinement.

You will enable dynamic adaption so that the solver periodically refines the mesh in the vicinity of the shocks as the iterations progress. The shocks are identified by their large pressure gradients.

Here 19996	Setting	Up Don	nain → A	dapt → I	Mark/Adapt	Cells \rightarrow	Gradient

Gradient Adaption				×
Options Image: Contours Manage Controls	Method Curvature Gradient Iso-Value Normalization Standard Scale Normalize Dynamic Interval 100	Gradients of Pressure Static Pressure Min 0 Coarsen Threshold 0.3	Max 0 Refine Threshold 0.7	•
Adapt	Mark	Compute Apply	Close Help	

a. Select **Gradient** from the **Method** group box.

The mesh adaption criterion can either be the gradient or the curvature (second gradient). Because strong shocks occur inside the nozzle, the gradient is used as the adaption criterion.

b. Select Scale from the Normalization group box.

Mesh adaption can be controlled by the raw (or standard) value of the gradient, the scaled value (by its average in the domain), or the normalized value (by its maximum in the domain). For dynamic mesh adaption, it is recommended that you use either the scaled or normalized value because the raw values will probably change strongly during the computation, which would necessitate a read-justment of the coarsen and refine thresholds. In this case, the scaled gradient is used.

- c. Enable **Dynamic** in the **Dynamic** group box.
- d. Enter 100 for the Interval.

For steady-state flows, it is sufficient to only seldomly adapt the mesh—in this case an interval of 100 iterations is chosen. For time-dependent flows, a considerably smaller interval must be used.

- e. Retain the default selection of Pressure... and Static Pressure from the Gradients of drop-down lists.
- f. Enter 0.3 for Coarsen Threshold.

g. Enter 0.7 for Refine Threshold.

As the refined regions of the mesh get larger, the coarsen and refine thresholds should get smaller. A coarsen threshold of 0.3 and a refine threshold of 0.7 result in a "medium" to "strong" mesh refinement in combination with the scaled gradient.

- h. Click **Apply** to store the information.
- i. Click the Controls... button to open the Mesh Adaption Controls dialog box.

💶 Mesh Adap	tion Controls	—
Options Refine	Zones Filter Text	Min Cell Volume (m3)
Coarsen	fluid	Min # of Cells
		0
		Max # of Cells
		20000 ≑
		Max Level of Refine
		2 🔹
		Volume Weight
		1
	OK Cancel Help	

- i. Retain the default selection of **fluid** in the **Zones** selection list.
- ii. Enter 20000 for Max # of Cells.

To restrict the mesh adaption, the maximum number of cells can be limited. If this limit is violated during the adaption, the coarsen and refine thresholds are adjusted to respect the maximum number of cells. Additional restrictions can be placed on the minimum cell volume, minimum number of cells, and maximum level of refinement.

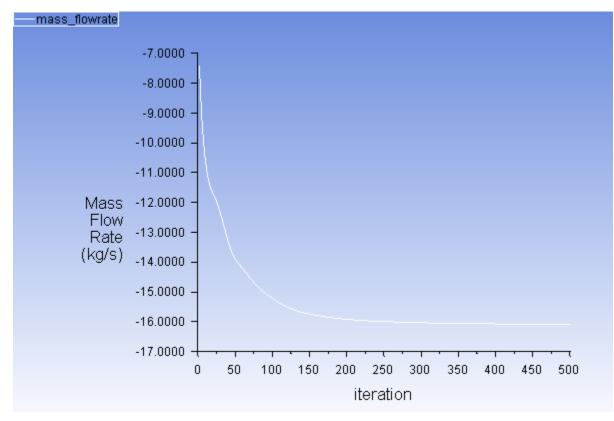
- iii. Click **OK** to save your settings and close the **Mesh Adaption Controls** dialog box.
- j. Click Close to close the Gradient Adaption dialog box.
- 8. Start the calculation by requesting 500 iterations.

Solving \rightarrow Run Calculation \rightarrow Advanced...

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

- a. Enter 500 for Number of Iterations.
- b. Click **Calculate** to start the steady flow simulation.





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

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

Note

When you write the case and data files at the same time, it does not matter whether you specify the file name with a .cas or .dat extension, as both will be saved.

- 10. Click **OK** in the **Question** dialog box to overwrite the existing file.
- 11. Review a mesh that resulted from the dynamic adaption performed during the computation.

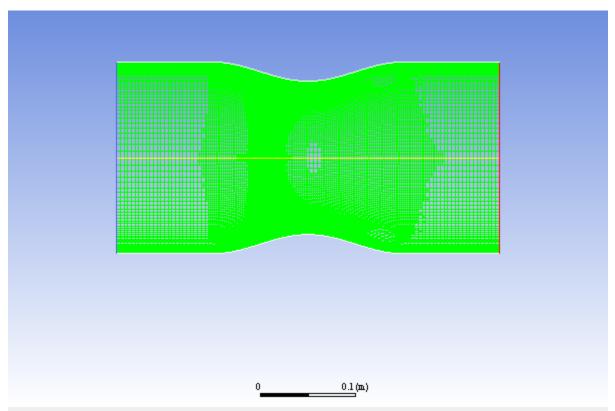
T Results →	Graphics \rightarrow Mesh	₫́Edit
-------------	-----------------------------	--------

💶 Mesh Display	,	
Options Nodes	Edge Type All	Surfaces Filter Text
Edges	Feature	default-interior
Faces	Outline	inlet
Partitions		lower-wall
Overset		outlet
Shrink Factor	Feature Angle	symmetry
0	20	
Outline	Interior	
Adjacency		New Surface 💌
	Di	isplay Colors Close Help

- a. Ensure that only the **Edges** option is enabled in the **Options** group box.
- b. Select Feature from the Edge Type list.
- c. Ensure that all of the items are selected from the **Surfaces** selection list.
- d. Click **Display** and close the **Mesh Display** dialog box.

The mesh after adaption is displayed in the graphics window (Figure 6.4: 2D Nozzle Mesh after Adaption (p. 287))





e. Zoom in using the middle mouse button to view aspects of your mesh.

Notice that the cells in the regions of high pressure gradients have been refined.

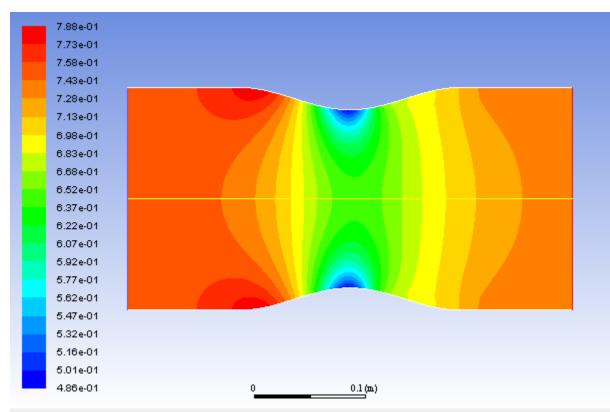
12. Display the steady flow contours of static pressure (Figure 6.5: Contours of Static Pressure (Steady Flow) (p. 288)).

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

Contours	
Options Filled Node Values	Contours of Pressure Static Pressure
Global Range Auto Range Clip to Range	Min Max 0 0
Draw ProfilesDraw Mesh	Surfaces Filter Text
Coloring Banded Smooth	inlet lower-wall outlet symmetry
Levels Setup	New Surface 🔻
	Display Compute Close Help

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

Figure 6.5: Contours of Static Pressure (Steady Flow)



The steady flow prediction in Figure 6.5: Contours of Static Pressure (Steady Flow) (p. 288) shows the expected pressure distribution, with low pressure near the nozzle throat.

13. Display the steady-flow velocity vectors (Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow) (p. 290)).

U Vectors	
Options Global Range Auto Range Clip to Range Auto Scale Draw Mesh	Vectors of Velocity
Style	Min Max 0 0 Surfaces Filter Text
50 0 🜩 Vector Options Custom Vectors	default-interior inlet lower-wall outlet symmetry
	New Surface Display Compute Close Help

Postprocessing \rightarrow Graphics \rightarrow Vectors \rightarrow Edit...

- a. Enter 50 under Scale.
- b. Click **Display** and close the **Vectors** dialog box.

The steady flow prediction shows the expected form, with a peak velocity of approximately 300 m/s through the nozzle.

You can zoom in on the wall in the expansion region of the nozzle to view the recirculation of the flow as shown in Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow) (p. 290).

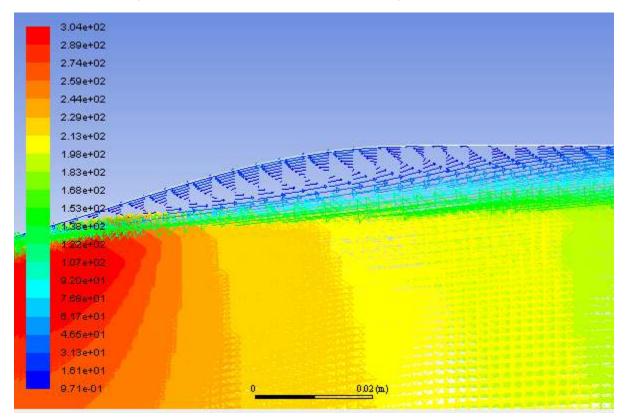


Figure 6.6: Velocity Vectors Showing Recirculation (Steady Flow)

14. Check the mass flux balance.

Important

Although the mass flow rate history indicates that the solution is converged, you should also check the mass flux throughout the domain to ensure that mass is being conserved.



Options Mass Flow Rate	Boundaries Filter Text	Results
 Total Heat Transfer Rate Radiation Heat Transfer Rate 	default-interior	
	inlet	16.10331527833818
	lower-wall	
	outlet	-16.10365265607839
	4 P	4
ave Output Parameter		Net Results (kg/s)
		-0.0003373777

- a. Retain the default selection of Mass Flow Rate.
- b. Select inlet and outlet in the Boundaries selection list.
- c. Click **Compute** and examine the values displayed in the dialog box.

Important

The net mass imbalance should be a small fraction (for example, 0.1%) of the total flux through the system. The imbalance is displayed in the lower right field under **Net Results**. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

d. Close the **Flux Reports** dialog box.

6.4.9. Enabling Time Dependence and Setting Transient Conditions

In this step you will define a transient flow by specifying a transient pressure condition for the nozzle.

1. Enable a time-dependent flow calculation.

Setting Up Physics → Solver → Transient

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

The pressure at the outlet is defined as a wave-shaped profile, and is described by the following equation:

$$p_{exit}(t) = 0.12\sin(\omega t) + \overline{p}_{exit}$$
(6.1)

where

 ω = circular frequency of transient pressure (rad/s)

 \overline{p}_{exit} = mean exit pressure (atm)

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

A user-defined function (pexit.c) has been written to define the equation (Equation 6.1 (p. 291)) required for the pressure profile.

Note

To input the value of Equation 6.1 (p. 291) in the correct units, the function pexit.c has to be written in SI units.

More details about user-defined functions can be found in the Fluent Customization Manual.

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

$\blacksquare User Defined \rightarrow User Defined \rightarrow Functions \rightarrow Interpreted...$

a. Enter pexit.c for Source File Name.

If the UDF source file is not in your working directory, then you must enter the entire directory path for **Source File Name** instead of just entering the file name.

b. Click Interpret.

The user-defined function has already been defined, but it must be compiled within ANSYS Fluent before it can be used in the solver.

- c. Close the Interpreted UDFs dialog box.
- 3. Define the transient boundary conditions at the nozzle exit (**outlet**).



💶 Pressure Out	let					×
Zone Name						
outlet						
Momentum	Thermal	Radiation	Species	DPM Multip	hase Potential	UDS
B	ackflow Refe	rence Frame	Absolute			•
	Gaug	je Pressure (atm)		udf transient_	pressure 💌
Backflow Direct	tion Specifica	tion Method	Normal to B	oundary		•
🗐 Average Pre	essure <mark>S</mark> pecif	ication	6			17
🔲 Target Mass	Flow Rate					
	Turbuler	ice				
	Specificat	ion Method	Intensity and	Viscosity Ratio		•
		Back	flow Turbuler	nt Intensity (%)	1.5	P
		Backf	low Turbulen	t Viscosity Ratio	10	P
Acoustic Way	e Model					
Off						
Non Refle	cting					
2 2 2 2		C	OK Cancel	Help		

- a. Select udf transient_pressure (the user-defined function) from the Gauge Pressure drop-down list.
- b. Click **OK** to close the **Pressure Outlet** dialog box.
- 4. Update the gradient adaption parameters for the transient case.

Setting Up Domain \rightarrow Adapt \rightarrow Mark/Adapt Cells \rightarrow Gradient...

a. Enter 10 for Interval in the Dynamic group box.

For the transient case, the mesh adaption will be done every 10 time steps.

- b. Enter 0.3 for Coarsen Threshold.
- c. Enter 0.7 for Refine Threshold.

The refine and coarsen thresholds have been changed during the steady-state computation to meet the limit of 20000 cells. Therefore, you must reset these parameters to their original values.

- d. Click Apply to store the values.
- e. Click Controls... to open the Mesh Adaption Controls dialog box.
 - i. Enter 8000 for Min # of Cells.
 - ii. Enter 30000 for Max # of Cells.

You must increase the maximum number of cells to try to avoid readjustment of the coarsen and refine thresholds. Additionally, you must limit the minimum number of cells to 8000, because you should not have a coarse mesh during the computation (the current mesh has approximately 20000 cells).

4

- iii. Click OK to close the Mesh Adaption Controls dialog box.
- f. Close the Gradient Adaption dialog box.

6.4.10. Specifying Solution Parameters for Transient Flow and Solving

1. Modify the **mass_flowrate_out-rfile** report file definition.

		Solution -	Monitors →	Report Files →	mass_flowrate_	_out-rfile 🖵	Edit
--	--	------------	------------	----------------	----------------	--------------	------

Edit Report File Jame	
mass_flowrate_out-rfile	
wallable Report Definitions [0/3]	Selected Report Definitions [0/1]
delta-time flow-time iters-per-timestep	Add>> < <remove< th=""></remove<>
Output File Base Name	New 💌 Edit
noz_uns.out Browse	
Full File Name:\solution_files\\noz_ss.out	
Get Data Every	
1 🔹 time-step 🔹	
T Cellin acob	

- a. Enter noz_uns.out for Output File Base Name.
- b. Select time-step from the Get Data Every drop-down list.
- c. Click **OK** to close the **Edit Report File** dialog box.
- 2. Modify the mass_flowrate_out-rplot report plot definition.

Solution \rightarrow Monitors \rightarrow Report Plots \rightarrow mass_flowrate_out-rplot $\stackrel{\Box}{\rightarrow}$ Edit...

Name mass flowrate out-rplot Available Report Definitions [0/0] Add>> Add>> Add>> Certea Report Definitions [0/1] (Add>> (<remove) (<remove) New Edit Get Data Every 1 © time-step VAxis Label time-step V-Axis Label time-step (Print to Console) Selected Report Definitions [0/1] New Edit</remove) </remove) 	mass flowrate_out-rplot Available Report Definitions [0/0] Add>> Add>> (Add>> (<remove)< th=""><th>Edit Report Plot</th><th></th></remove)<>	Edit Report Plot	
Available Report Definitions [0/0]	Available Report Definitions [0/0]	Name	
Add>> <	Add>> Add>> < <remove< td=""> Options Plot Window 2 Curves Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label time-step Image: Print to Console</remove<>	mass_flowrate_out-rplot	
Add>> < <remove <="" td=""> Options Plot Window 2 Curves Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate</remove>	Add>> < <remove <="" td=""> Options Plot Window 2 Curves Axes Get Data Every 1 1 time-step V-Axis Label time-step V-Axis Label time-step V-Axis Label Mass Flow Rate V Print to Console</remove>	Available Report Definitions [0/0]	Selected Report Definitions [0/1]
Options Plot Window 2 Curves Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Time-step	Options Plot Window 2		mass_flowrate_out
Options Plot Window 2 Curves Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate	Options Plot Window 2		
Options Plot Window 2 Curves Axes Get Data Every 1 Curves Axes Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate	Options Plot Window 2		
Plot Window 2 Curves Axes Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate	Plot Window 2 Curves Axes Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate Print to Console		< <kempve< td=""></kempve<>
Plot Window 2 Curves Axes Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate	Plot Window 2 Curves, Axes Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate V Print to Console		
Plot Window 2 Curves Axes Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate	Plot Window 2 Curves Axes Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate Print to Console		
Get Data Every 1 0 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate	Get Data Every 1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate V Print to Console		New TEdit
1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate	1 time-step Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate V Print to Console	2 Curves Axes	
Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate	Plot Title mass_flowrate_out-rplot X-Axis Label time-step Y-Axis Label Mass Flow Rate V Print to Console	Get Data Every	
X-Axis Label time-step Y-Axis Label Mass Flow Rate	X-Axis Label time-step Y-Axis Label Mass Flow Rate Print to Console	1 🔄 time-step 🔹	
X-Axis Label time-step Y-Axis Label Mass Flow Rate	X-Axis Label time-step Y-Axis Label Mass Flow Rate Print to Console	Plot Title mass_flowrate_out-rplot	
Y-Axis Label Mass Flow Rate	Y-Axis Label Mass Flow Rate		
Print to Console			
	OK Cancel Help	V Print to Console	
	OK Cancel Help	n - Area Maria - San Area I	

a. For Get Data Every, retain the value of 1 and select time-step from the drop-down list.

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

- b. Select time-step from the X-Axis Label drop-down list.
- c. Click **OK** to close the **Edit Report File** dialog box.
- 3. Save the transient solution case file (noz_uns.cas.gz).

File \rightarrow Write \rightarrow Case...

4. Modify the plotting of residuals.



- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Ensure none is selected from the Convergence Criterion drop-down list.
- c. Set the **Iterations to Plot** to 100.
- d. Click **OK** to close the **Residual Monitors** dialog box.
- 5. Define the time step parameters.

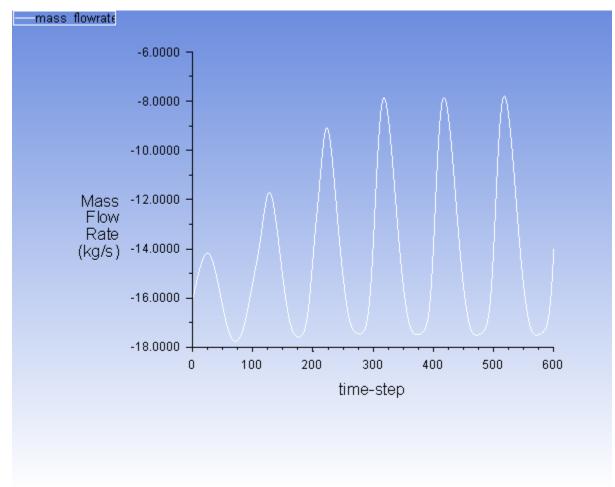
The selection of the time step is critical for accurate time-dependent flow predictions. Using a time step of 2.85596 x 10^{-5} seconds, 100 time steps are required for one pressure cycle. The pressure cycle begins and ends with the initial pressure at the nozzle exit.

Check Case	Preview Mesh Motion
ime Stepping Method	Time Step Size (s)
Fixed 🔻	2.85596e-5
Settings	Number of Time Steps
	600 🗘
Options	
Extrapolate Variable	S
Data Sampling for T	ime Statistics
Sampling Interval	
1	Sampling Options
Time Sample	ed (s) 0
Solid Time Step	
User Specified	
Automatic	
0	
	- Depending Tabanal
ax Iterations/Time Ste	
rofile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	

- a. Enter 2.85596e-5 s for Time Step Size.
- b. Enter 600 for Number of Time Steps.
- c. Enter 10 for Max Iterations/Time Step.
- d. Click **Calculate** to start the transient simulation.

By requesting 600 time steps, you are asking ANSYS Fluent to compute six pressure cycles. The mass flow rate history is shown in Figure 6.7: Mass Flow Rate History (Transient Flow) (p. 297).





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

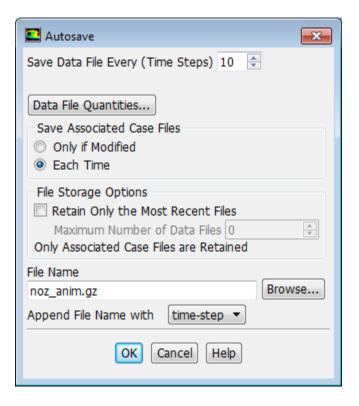
File → Write → Case & Data...

6.4.11. Saving and Postprocessing Time-Dependent Data Sets

At this point, the solution has reached a time-periodic state. To study how the flow changes within a single pressure cycle, you will now continue the solution for 100 more time steps. You will use ANSYS Fluent's solution animation feature to save contour plots of pressure and Mach number at each time step, and the autosave feature to save case and data files every 10 time steps. After the calculation is complete, you will use the solution animation playback feature to view the animated pressure and Mach number plots over time.

1. Request the saving of case and data files every 10 time steps.





- a. Enter 10 for Save Data File Every.
- b. Select Each Time for Save Associated Case Files.
- c. Retain the default selection of time-step from the Append File Name with drop-down list.
- d. Enter noz_anim.gz for File Name.

When ANSYS Fluent saves a file, it will append the time step value to the file name prefix (noz_anim). The standard extensions (.cas and .dat) will also be appended. By adding the optional extension .gz to the end of the file name, you instruct ANSYS Fluent to save the case and data files in compressed format. This will yield file names of the form noz_anim-1-00640.cas.gz and noz_anim-1-00640.dat.gz, where 00640 is the time step number.

e. Click OK to close the Autosave dialog box.

Tip

If you have constraints on disk space, you can restrict the number of files saved by ANSYS Fluent by enabling the **Retain Only the Most Recent Files** option and setting the **Maximum Number of Data Files** to a nonzero number.

2. Create an animation definition for the nozzle pressure contour plot.

Solving \rightarrow Activities \rightarrow Create \rightarrow Solution Animations...

Animation Definition			
Name: pressure			
Record after every	1 🗧 Time Step 🔻		
Storage Type	In Memory		
Storage Directory	sible/solution_files		
Window Id	3		
Animation Object	=		
residuals			
contour-1 surf-mon-1-pset			
sun-mon-1-pset			
New Object 🔻 Edit Object			
Save	Close Help		

- a. Enter pressure for the Name.
- b. Select Time Step for Record after every.

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

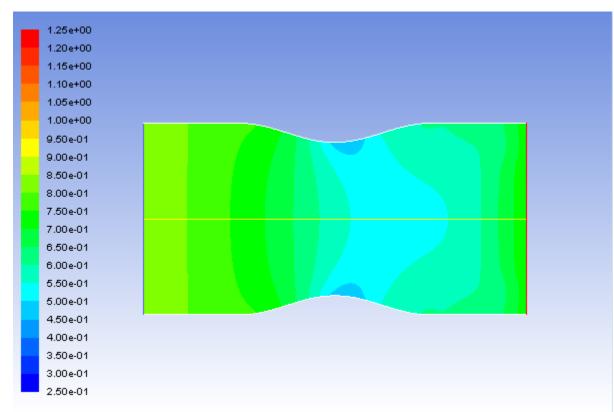
c. Select In Memory from the Storage Type drop-down.

The **In Memory** option is acceptable for a small 2D case such as this. For larger 2D or 3D cases, saving animation files with either the **Metafile** or **PPM Image** option is preferable, to avoid using too much of your machine's memory.

- d. Enter **3** for the **Window Id**.
- e. Click New Object and select Contours... from the drop-down list to open the associated dialog box.

Contours			— ×
Contour Name			
contour-1			
Options	Contours of		
Filled	Pressure		
Node Values	Static Pressure		•
Global Range Auto Range	Min (atm)	Max (atm)	
Clip to Range	0.25	1.25	
Draw Profiles	Surfaces Filter Text		x
	default-interior inlet		
Coloring	lower-wall		
Banded	outlet		
Smooth	symmetry		
Colormap Options	zone-surface-5		
colornap options	New Surface 🔻		
Save/Display Compute Close Help			

- i. Select **Pressure...** and **Static Pressure** from the **Contours of** drop-down lists.
- ii. Ensure that **Filled** is enabled in the **Options** group box.
- iii. Disable Auto Range.
- iv. Enter 0.25 atm for Min and 1.25 atm for Max.
- v. Click Save/Display and close the Contours dialog box.



vi. Figure 6.8: Pressure Contours at t=0.017136 s

- f. Ensure **contour-1** is selected in the **Animation Object** group box.
- g. Click Save and close the Animation Definition dialog box.
- 3. Create an animation definition for the Mach number contour plot.

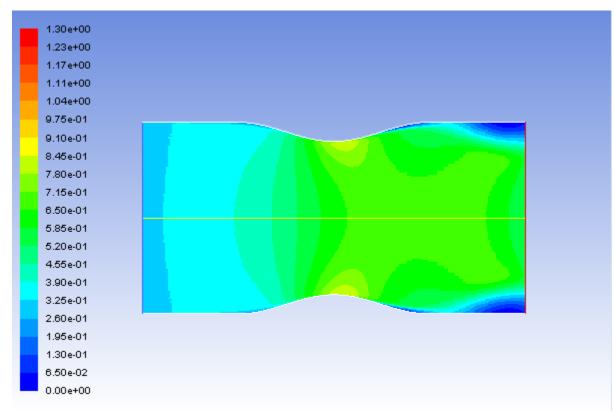
Solving \rightarrow Activities \rightarrow Create \rightarrow Solution Animations...

Animation Definition			
Name: mach-numbe	r		
Record after every	1 🗧 Time Step 🔻		
Storage Type	In Memory		
Storage Directory	sible/solution_files		
Window Id	4		
Animation Object	=		
residuals contour-1			
contour-2			
surf-mon-1-pset			
New Object Edit Object			
Save Close Help			

- a. Enter **mach-number** for the **Name**.
- b. Select Time Step for Record after every.

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

- c. Ensure that In Memory is selected from the Storage Type drop-down.
- d. Enter 4 for the Window Id.
- e. Click New Object and select Contours... from the drop-down list to open the associated dialog box.
 - i. Select Velocity... and Mach Number from the Contours of drop-down lists.
 - ii. Ensure that **Filled** is enabled in the **Options** group box.
 - iii. Disable Auto Range.
 - iv. Enter 0.00 for Min and 1.30 for Max.
 - v. Click Save/Display and close the Contours dialog box.



vi. Figure 6.9: Mach Number Contours at t=0.017136 s

- f. Ensure contour-2 is selected in the Animation Object group box.
- g. Click **Save** and close the **Animation Definition** dialog box.
- 4. Continue the calculation by requesting 100 time steps.

By requesting 100 time steps, you will march the solution through an additional 0.0028 seconds, or roughly one pressure cycle.

With the autosave and animation features active (as defined previously), the case and data files will be saved approximately every 0.00028 seconds of the solution time; animation files will be saved every 0.00028 seconds of the solution time.



Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method	Time Step Size (s)
Fixed 🔹	2.85596e-05
Settings	Number of Time Steps
	100
Options	
Extrapolate Variables	Statistics
Sampling Interval	100000
	Sampling Options
Time Sampled (s	0
Max Iterations/Time Step	Reporting Interval
Profile Update Interval	
1	
Data File Quantities	Acoustic Signals
Calculate	
Help	

Enter 100 for Number of Time Steps and click Calculate.

When the calculation finishes, you will have ten pairs of case and data files and there will be 100 pairs of contour plots stored in memory. In the next few steps, you will play back the animation sequences and examine the results at several time steps after reading in pairs of newly saved case and data files.

5. Play the animation of the pressure contours.



Playback	×
Playback Playback Mode Play Once Start Frame Increment End Frame 1 1 1 1 100 1 Frame I I I I I I I I I I I I I I I I I I I	Animation Sequences Sequences pressure mach-number
Slow Replay Speed Fast	Delete Delete All
Write/Record Format Animation Frames	Picture Options
Write Read Close	Help

a. Retain the default selection of pressure in the Sequences selection list.

Ensure that tab window 4 is open in the graphics window.

- b. Click the play button (the second from the right in the group of buttons in the **Playback** group box).
- c. Close the **Playback** dialog box.

Examples of pressure contours at t=0.017993 s (the 630th time step) and t=0.019135 s (the 670th time step) are shown in Figure 6.10: Pressure Contours at t=0.017993 s(p. 306) and Figure 6.11: Pressure Contours at t=0.019135 s (p. 306).

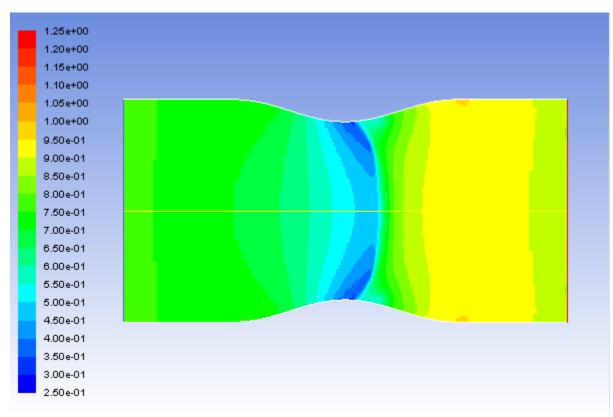
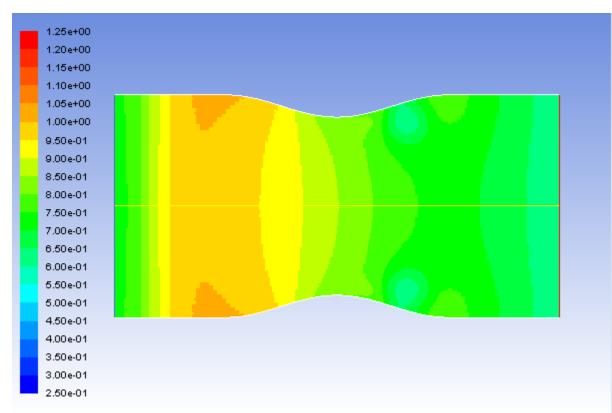


Figure 6.10: Pressure Contours at t=0.017993 s





6. In a similar manner to steps 4 and 5, select the appropriate active window and animation sequence name for the Mach number contours.

Examples of Mach number contours at t=0.017993 s and t=0.019135 s are shown in Figure 6.12: Mach Number Contours at t=0.017993 s (p. 307) and Figure 6.13: Mach Number Contours at t=0.019135 s(p. 308).

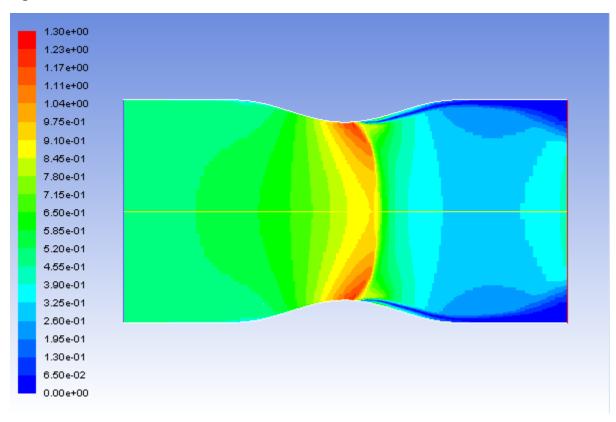


Figure 6.12: Mach Number Contours at t=0.017993 s

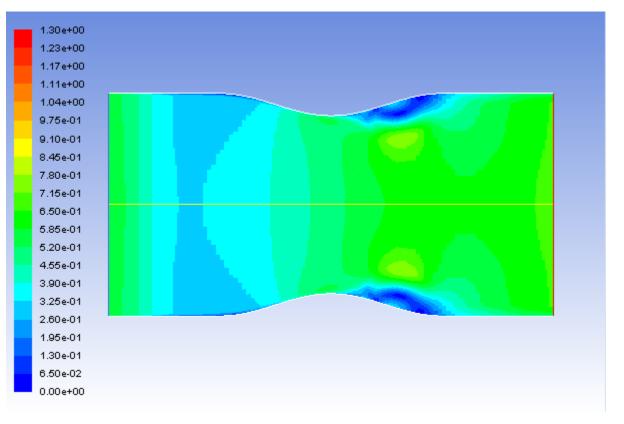


Figure 6.13: Mach Number Contours at t=0.019135 s

Тір

ANSYS Fluent gives you the option of exporting an animation as an MPEG file or as a series of files in any of the hardcopy formats available in the **Save Picture** dialog box (including TIFF and PostScript).

To save an MPEG file, select **MPEG** from the **Write/Record Format** drop-down list in the **Playback** dialog box and then click the **Write** button. The MPEG file will be saved in your working folder. You can view the MPEG movie using an MPEG player (for example, Windows Media Player or another MPEG movie player).

To save a series of TIFF, PostScript, or other hardcopy files, select **Picture Frames** in the **Write/Record Format** drop-down list in the **Playback** dialog box. Click the **Picture Op-tions...** button to open the **Save Picture** dialog box and set the appropriate parameters for saving the hardcopy files. Click **Apply** in the **Save Picture** dialog box to save your modified settings. Click **Save...** to select a directory in which to save the files. In the **Playback** dialog box, click the **Write** button. ANSYS Fluent will replay the animation, saving each frame to a separate file in your working folder.

If you want to view the solution animation in a later ANSYS Fluent session, you can select **Animation Frames** as the **Write/Record Format** and click **Write**.

Warning

Because the solution animation was stored in memory, it will be lost if you exit ANSYS Fluent without saving it in one of the formats described previously. Note that only the animation-frame format can be read back into the **Playback** dialog box for display in a later ANSYS Fluent session.

- 7. Read the case and data files for the 660th time step (**noz_anim-1-00660.cas.gz** and **noz_an-im-1-00660.dat.gz**) into ANSYS Fluent.
- 8. Plot vectors at t=0.018849 s (Figure 6.14: Velocity Vectors at t=0.018849 s (p. 310)).

Vectors	
Options Ø Global Range	Vectors of Velocity
 Auto Range Clip to Range 	Color by Velocity
Auto Scale Draw Mesh	Velocity Magnitude Min (m/s) Max (m/s) Max (m/s)
Style	0.3684946 262.8595
Scale Skip	Surfaces Filter Text
50 50 Vector Options Custom Vectors	inlet lower-wall outlet symmetry
	New Surface Display Compute Close Help

Postprocessing \rightarrow Graphics \rightarrow Vectors \rightarrow Edit...

- a. Ensure Auto Scale is enabled under Options.
- b. Enter 50 under Scale.
- c. Enter 50 under Skip.
- d. Click **Display** and close the **Vectors** dialog box.

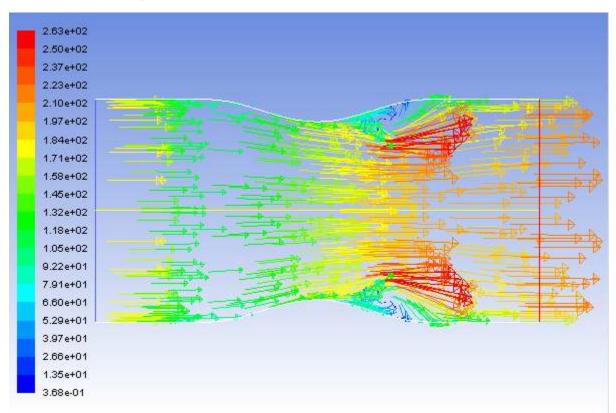


Figure 6.14: Velocity Vectors at t=0.018849 s

The transient flow prediction in Figure 6.14: Velocity Vectors at t=0.018849 s(p. 310) shows the expected form, with peak velocity of approximately 260 m/s through the nozzle at t=0.018849 seconds.

9. In a similar manner to steps 7 and 8, read the case and data files saved for other time steps of interest and display the vectors.

6.5. Summary

In this tutorial, you modeled the transient flow of air through a nozzle. In doing so, you learned how to:

- generate a steady-state solution as an initial condition for the transient case.
- set solution parameters for implicit time-stepping and apply a user-defined transient pressure profile at the outlet.
- use mesh adaption to refine the mesh in areas with high pressure gradients to better capture the shocks.
- automatically save solution information as the transient calculation proceeds.
- create and view solution animations of the transient flow.

6.6. Further Improvements

This tutorial guides you through the steps to generate a second-order solution. You may be able to increase the accuracy of the solution even further by using an appropriate higher-order discretization scheme and by adapting the mesh further. Mesh adaption can also ensure that the solution is independ-

ent of the mesh. These steps are demonstrated in Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121).