# Chapter 1: Fluid Flow and Heat Transfer in a Mixing Elbow

This tutorial is divided into the following sections:

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

# 1.1. Introduction

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

This tutorial demonstrates how to do the following:

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

# **1.2. Prerequisites**

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

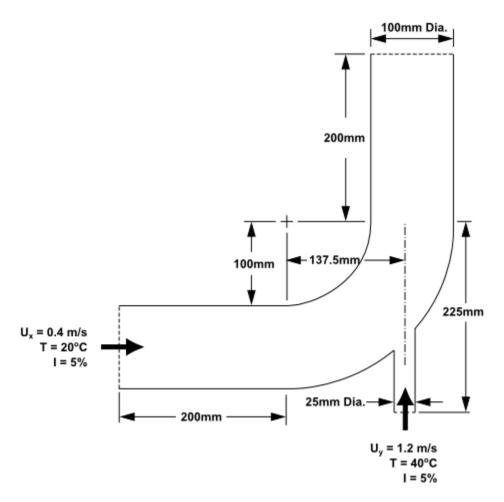
# **1.3. Problem Description**

The problem to be considered is shown schematically in Figure 1.1: Problem Specification (p. 2). A cold fluid at 20° C flows into the pipe through a large inlet, and mixes with a warmer fluid at 40° C that enters through a smaller inlet located at the elbow. The pipe dimensions are in inches and the fluid properties and boundary conditions are given in SI units. The Reynolds number for the flow at the larger inlet is 50,800, so a turbulent flow model will be required.

### Note:

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

### Figure 1.1: Problem Specification



# 1.4. Setup and Solution

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

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

- 1.4.1. Preparation
- 1.4.2. Launching ANSYS Fluent
- 1.4.3. Reading the Mesh
- 1.4.4. Setting Up Domain
- 1.4.5. Setting Up Physics
- 1.4.6. Solving
- 1.4.7. Displaying the Preliminary Solution
- 1.4.8. Adapting the Mesh

# 1.4.1. Preparation

- 1. Download the introduction.zip file here.
- 2. Unzip introduction.zip to your working directory.
- 3. The elbow.msh can be found in the folder.

### Note:

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

# 1.4.2. Launching ANSYS Fluent

1. From the Windows **Start** menu, select **Start > ANSYS 2020 R1 > Fluent 2020 R1** to start Fluent Launcher.

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

Fluent Launcher 2020 R1	-	-	o ×
Fluent Launcher			ANSYS
Meshing	Simulate a wide range of industrial applications using the general-purpo and post-processing capabilities of ANSYS Fluent. Get Started With Dimension	se seti	ıp, solve,
Solution	Case Case and Data O 2D		
Icing	Mesh Journal O 3D		
	Options       Precent Files       Pouble Precision       Poisplay Mesh After R       Load ACT       Start Server	-	
	Parallel (Local Machine Solver Processes	e) 4	¢
	Solver GPGPUs per Mach	ine 0	\$
✓ Show More Options ✓ Show	ow Learning Resources		
	Start Reset Cancel Help 🗸		

- 2. Ensure that the proper options are enabled.
  - a. Select **3D** from the **Dimension** list by clicking the radio button or the text.
  - b. Ensure that the **Double Precision** option is selected.
  - c. Ensure that the **Display Mesh After Reading** option is enabled.
  - d. Set Processes to 4 under the Parallel (local Machine).

### Note:

Fluent will retain your preferences for future sessions.

- 3. Set the working folder to the one created when you unzipped introduction.zip.
  - a. Click the **Show More Options** button to reveal additional options.
  - b. Enter the path to your working folder for **Working Directory** by double-clicking the text box and typing.

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

Fluent Launcher 2020 R1					_		$\times$
Fluent Launcher						٨N	SYS
Meshing Solution Icing	and post-p	orocessing ca ted With Case Mesh	of industrial ap spabilities of AN Case and E Journal	Dimension Data O 2D	n fter Read achine)	ding	olve,
∽ Show Fewer Options ➤ Show Fewer Options General Options Parallel Settings	ow Learnin Remote	ng Resource	Environment				
Pre/Post Only	Renote	Scheduler	Environment				
Working Directory							
						•	2
Fluent Root Path					•	· 📄	5

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

0										- 0 ×
Dealer Physics	ks incr-Ocfined	Solution Re-	selts Ve	~ P	analiel Desig				Q, quai terra (a	(H) 🔅 🖬 🖊 🕅 🕅
ом. 🛞 🛞 2	Transform _ U_ Separat	te . mi Desctivate	Chapterd	Interfaces Interfaces Interfaces Interfaces	Hesh Models C Dynamic Mesh., 24 Mixing Planes.,	Turbo Midde Insubie I Turbo Topolo Piel Turbo Crosso	Befine / Coorsen	Barfaie + Drote A Monope		
Outline View	<ul> <li>Tank Pa</li> </ul>	ige .								
Fibr Ted	General	•		Got Started	with Calif		to these	What's a	kow Release Notes	Latert News AVG/S News Center
setup     B desnal     B desnal     B desnal     Solution     Solution     Constant     Con	Solar Type B In O In Type B In D In	eley ennum-Record ennum-Record (○ For endy andext					Loanning Resources Tutural Head How and Heat Tuturale in Nooking Datamal Composable Dang a Bringh Rutaring Releven Mooking Social Transport and Your More Professional States (Second States) And States (Second States) And States (Second States) And States (Second States) And States)	a Hong (bev Flan a Flane	Volese Volese AVD'S Ruet Vor VO'S Ruet Mas	apace 34th dustion ac Heating for CP3 Simulations a Separat Post Tenk Design and Manufe
			De La		page at skolup <b>1 [2 C C C</b> interestations - defen	4.	V		2	(a - ) ; 0
		_	C18	woup accept #	the Le DI'ANDYSDER	Flownt-Tutoria	ala-Jaro#1/01_Latroductio	n/coessup-foues	t-watbjzcdos-13000.Bet	

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

### 1.4.3. Reading the Mesh

1. Read the mesh file elbow.msh.

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

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

Select File				?	$\times$
Look in: C:\Tutorials\introduction	G	O	0	<b>6</b> (	: =
Ny Computer       Name       Size       Type       Date M         Documents       elbow.msh       2.59 MiB msle       1M					
Mesh File Files of type: Mesh Files (*.msh* *.MSH* ) Filter String				22	OK Cancel Filter
✓ Display Mesh After Reading					//

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

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

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

### Extra:

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

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

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

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

### **Table 1.1: View Manipulation Instructions**

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

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

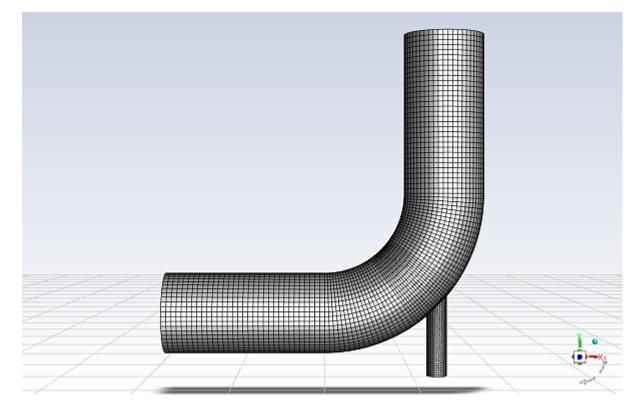
mouse button settings by clicking

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

icon, <sup>22</sup>. This will display the length scale ruler near the bottom of the graphics window.

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

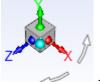
- 2. Manipulate the mesh display using the axis triad to obtain a front view as shown in Figure 1.2: The Hexahedral Mesh for the Mixing Elbow (p. 9).
  - a. Click the z-axis.
  - b. Clicking the **Fit to Window** icon, , will cause the object to fit exactly and be centered in the window.
  - c. Figure 1.2: The Hexahedral Mesh for the Mixing Elbow



### Extra:

axis triad

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



as follows:

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

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

# 1.4.4. Setting Up Domain

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



1. Check the mesh.



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

```
Domain Extents:
    x-coordinate: min (m) = -8.000000e+00, max (m) = 8.000000e+00
    y-coordinate: min (m) = -9.134634e+00, max (m) = 8.000000e+00
    z-coordinate: min (m) = 0.000000e+00, max (m) = 2.000000e+00
Volume statistics:
    minimum volume (m3): 5.098304e-04
    maximum volume (m3): 2.330736e-02
        total volume (m3): 1.607154e+02
Face area statistics:
    minimum face area (m2): 4.865882e-03
    maximum face area (m2): 1.017924e-01
Checking mesh......
Done.
```

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

### Note:

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

### 2. Scale the mesh.

Scaling
Scaling
<ul> <li>Convert Units</li> <li>Specify Scaling Factors</li> <li>Mesh Was Created In         <ul> <li>in</li> <li>Scaling Factors</li> <li>X 0.0254</li> <li>Y 0.0254</li> <li>Z 0.0254</li> <li>Scale</li> <li>Unscale</li> </ul> </li> </ul>

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

### Warning:

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

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

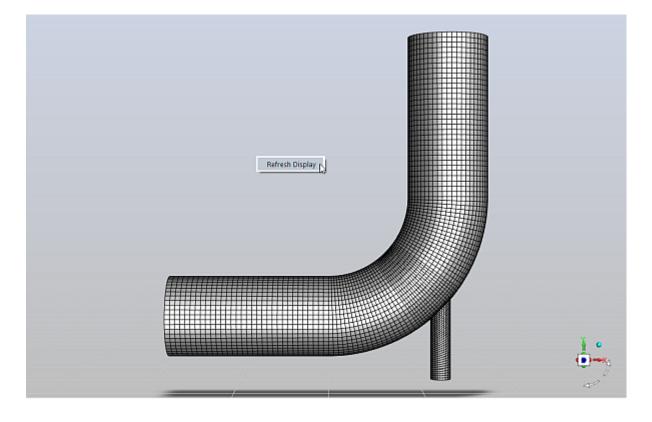
- d. Select **in** from the **View Length Unit In** drop-down list to set inches as the working unit for length.
- e. Confirm that the domain extents are as shown in the previous dialog box.
- f. Close the Scale Mesh dialog box.

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

### Note:

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

3. Right click in the graphics window and select Refresh Display



- 4. Clicking the **Fit to Window** icon, , will cause the object to fit exactly and be centered in the window.
- 5. Check the mesh.

# **Domain** $\rightarrow$ Mesh $\rightarrow$ Check $\rightarrow$ Perform Mesh Check

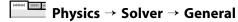
### Note:

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

### 1.4.5. Setting Up Physics

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

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



Task Page		<
General		?
Mesh		
Scale	Check Report Quality	
Display	Jnits	
Solver		
Туре	Velocity Formulation	
Pressure-Based	<ul> <li>Absolute</li> </ul>	
O Density-Based	Relative	
<ul> <li>Steady</li> <li>Transient</li> </ul>		
Gravity		

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

	M	lodels	
	📶 Radiation	🙊 Multiphase	<b>⊿</b> Structure
Energy	∦≓ Heat Exchanger	••• Species	Coustics
	🕼 Viscous	🌽 Discrete Phase	👓 More 🚽

### Note:

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

a. Enable heat transfer by activating the energy equation.

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

Physics  $\rightarrow$  Models  $\rightarrow$  Energy

### Note:

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

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

Physics  $\rightarrow$  Models  $\rightarrow$  Viscous...

E Viscous Model	×		
Model	Model Constants		
Inviscid	Alpha*_inf		
🔿 Laminar	1		
O Spalart-Allmaras (1 eqn)	Alpha_inf		
k-epsilon (2 eqn)	0.52		
<ul> <li>k-omega (2 eqn)</li> <li>Transition k-kl-omega (3 eqn)</li> <li>Transition SST (4 eqn)</li> <li>Reynolds Stress (7 eqn)</li> <li>Scale-Adaptive Simulation (SAS)</li> <li>Detached Eddy Simulation (DES)</li> </ul>	Beta*_inf		
	0.09		
	a1		
	0.31		
	Beta_i (Inner)		
Large Eddy Simulation (LES)	0.075		
	Beta_i (Outer)		
k-omega Model			
Standard	User-Defined Functions		
	Turbulent Viscosity		
SST	none 💌		
	Prandtl Numbers		
k-omega Options	Energy Prandtl Number		
Low-Re Corrections	none 🔻		
Options	Wall Prandtl Number		
Viscous Heating	none 💌		
Curvature Correction			
Production Kato-Launder			
Production Limiter			
Intermittency Transition Model			
ОКС	ancel Help		

- i. Retain the default selection of **k-omega** from the **Model** list.
- ii. Retain the default selection of **SST** in the **k-omega Model** group box.
- iii. Click **OK** to accept all the other default settings and close the **Viscous Model** dialog box.

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

### Note:

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

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



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

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

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

lame	M	laterial Type		aterials by
water-liquid		fluid	🚽 🖲 Nam	
Chemical Formula	F	luent Fluid Materials	O Che	mical Formula
h2o <l></l>	(	water-liquid (h2o <l>)</l>	- Flue	ent Database
	M	lixture	Ucor-D	efined Database.
		none	-	enneu batabase.
	Properties			
	Density (kg/m	3) constant	Edit	
		998.2		
	Cp (Specific Heat) (j/kg-	k) constant	▼ Edit	
		4182		
	Thermal Conductivity (w/m-	k) constant	▼ Edit	
		0.6		
	Viscosity (kg/m-	s) constant	▼ Edit 🖕	

- b. Click the Fluent Database... button to access pre-defined materials.
- c. Select **water-liquid** (h2o < l >) from the materials list and click **Copy**, then close the **Fluent Database...** dialog box.

Fluent Database Materials			$\times$
Fluent Fluid Materials [1/563]	= = .	Material Type fluid	*
vinyl-trichlorosilane (sicl3ch2ch) vinylidene-chloride (ch2ccl2)	<b>^</b>	Order Materials by	
water-liquid (h2o <l>)</l>		Name	
water-vapor (h2o)		O Chemical Formu	la
wood-volatiles (wood_vol)			
	Ψ		
Copy Materials from Case Delete			
Properties			
Density (kg/m3)	constant	•	View
	998.2		
Cp (Specific Heat) (j/kg-k)	constant	•	View
	4182		
Thermal Conductivity (w/m-k)	constant	-	View
	0.6		
Viscosity (kg/m-s)	constant	•	View
	0.001003		
	· ·		
New Edit	Save Copy Clos	Help	

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

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

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

Zones
Cell Zones
Η Boundaries
Profiles

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

 $\blacksquare Physics \rightarrow Zones \rightarrow Cell Zones$ 

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

Cell Zone Conditions	?
Zone Filter Text	F,
fluid	
Phase Type ID -1 Edit Copy Profiles Parameters Display Mesh	
Porous Formulation     Superficial Velocity     Physical Velocity	

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

Pluid Fluid						×
Zone Name [fluid Material Name water-liquid  Frame Motion 3D Fan Zone Source Terms Mesh Motion Laminar Zone Fixed Values						
Porous Zone     Reference Frame Mesh Motion Porous Zone 3D Fan Zone     Rotation-Axis Origin	Em	bedded LES	Reaction	Source Terms	Fixed Values	Multiphase
X (in) 0 Y (in) 0 Z (in) 0	) • • • Cane	x0 Y0 Z1				

### Note:

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

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



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

Physics  $\rightarrow$  Zones  $\rightarrow$  Boundaries

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

Boundary Conditions	)
Zone Filter Text   Image: Substrain of the state of th	
Phase       Type       ID         mixture       ID       -1         Edit       Copy       Profiles         Parameters       Operating Conditions         Display Mesh       Periodic Conditions         Highlight Zone       ID	

### Note:

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

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

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

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

### Tip:

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

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

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

📕 Velocity l	nlet						$\times$
Zone Name							
velocity-inlet	t-5						
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
	Velocity Spe	ecification Me	ethod Magni	tude, Norma	l to Boundary		•
		Reference Fi	rame Absolı	ute			•
	Velocity	y Magnitude (	(m/s) 0.4				•
Supersonic/	Initial Gauge	Pressure (pa	ascal) 0				•
	Turbuk	ence					
	Spec	cification Met	hod Intensit	y and Hydrau	ulic Diameter		•
	Turbule	ent Intensity	(%) 5				•
Hydraulic Diameter (in) 4					•		
			OK Can	Help	)		

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

The hydraulic diameter  $D_h$  is defined as:

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

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

vii. Click the **Thermal** tab.

🖪 Velocity In	let						$\times$
Zone Name							
velocity-inlet-	-5						
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Temperature	e (k) 293.15				-		
			OK Can	cel Help			

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

ix. Click **OK** to close the **Velocity Inlet** dialog box.

### Note:

You can also access the **Velocity Inlet** dialog box by double-clicking the **Setup/Boundary Conditions/velocity-inlet-5** tree item.

c. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-6**), using the values in the following table:

Setting	Value
Velocity Specification Method	Magnitude, Normal to Boundary
Velocity Magnitude	1.2 [m/s]
Specification Method	Intensity and Hydraulic Diameter
Turbulent Intensity	5 [%]
Hydraulic Diameter	1 [inch]
Temperature	313.15 [K]

d. Double-click **pressure-outlet-7** in the **Zone** selection list and set the boundary conditions at the outlet, as shown in the following figure.

Pressure	Outlet						$\times$
Zone Name							
pressure-ou	tlet-7						
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
	Backflow Re	ference Frar	ne Absolute	9			-
	Gauge Pr	essure (paso	al) 0				•
Pressure Profile Multiplier			lier 1				-
Backflow Direction Specification Method Normal to Boundary						•	
Bac	kflow Pressur	re Specificati	on Total Pr	essure			•
Radial E	quilibrium Pr	essure Distri	bution				
Average	e Pressure Sp	ecification					
🗌 Target I	Mass Flow Ra	ite					
Turbu	lence						
	Specifi	cation Metho	dIntensity	and Hydraul	ic Diameter		•
Backflow Turbulent Intensity (%) 5				-			
Backf	low Hydraulic	Diameter (ir	n)[4				-
		l	OK Can	cel Help			

### Note:

- You do not need to set a backflow temperature in this case (in the **Thermal** tab) because the material properties are not functions of temperature. If they were, a flow-weighted average of the inlet conditions would be a good starting value.
- ANSYS Fluent will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.
- e. For the wall of the pipe (wall), retain the default value of 0 W/m<sup>2</sup> for Heat Flux in the Thermal tab.

💌 Wall									×
Zone Name									
wall									
Adjacent Cell	Zone					_			
fluid									
Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential	Structure
Wall Motion	n	Motion							
<ul> <li>Station</li> <li>Moving</li> </ul>	-	✓ Relativ	ve to Adjac	ent Cell	Zone				
Shear Cond	lition								
No Slip									
	ed Shear	clout							
	arity Coeffi Joni Stress								
Wall Rough	iness								
Roughness	Height (in)	0					-		
Roughnes	s Constant	0.5					•		
			O	c Car	ncel Help				

# 1.4.6. Solving

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

### Note:

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

- 1. Select a solver scheme.
  - a. In the Solution ribbon tab, click Methods... (Solution group box).

Solution





Solution Methods	?
Pressure-Velocity Coupling	
Scheme	
Coupled	•
Spatial Discretization	
Gradient	<b>^</b>
Least Squares Cell Based	<b>_</b>
Pressure	
Second Order	<b>_</b>
Momentum	
Second Order Upwind	<b>_</b>
Turbulent Kinetic Energy	
First Order Upwind	<b>_</b>
Turbulent Dissipation Rate	
First Order Upwind	•
Energy	Υ
Transient Formulation	
•	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
✓ Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	

- b. Retain the default selections.
- 2. Enable the plotting of residuals during the calculation.
  - a. In the Solution ribbon tab, click Residuals... (Reports group box).

R	eports	
🖄 Residuals	)¥ Conver	gence
📉 Definitions 🖕	File	∫ Plot

### **Solution** $\rightarrow$ Reports $\rightarrow$ Residuals...

Residual Monitors					$\times$	
Options	Equations					
✓ Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria	^	
✓ Plot	continuity	✓	<b>v</b>	0.001		
	x-velocity	$\checkmark$	✓	0.001		
Curves Axes	y-velocity	~	$\checkmark$	0.001		
Iterations to Plot	z-velocity	✓	$\checkmark$	0.001		
	energy	~	✓	1e-06		
Iterations to Store	k	✓	$\checkmark$	0.001		
1000 🗘	omega	~	✓	0.001	¥	
Convergence Conditions  Show Advanced Options						
OK Plot Cancel Help						

### Note:

You can also access the **Residual Monitors** dialog box by double-clicking the **Solution/Monitors/Residual** tree item.

- b. Ensure that **Plot** is enabled in the **Options** group box.
- c. Retain the default value of 0.001 for the **Absolute Criteria** of **continuity**.

d. Click **OK** to close the **Residual Monitors** dialog box.

### Note:

By default, the residuals of all of the equations solved for the physical models enabled for your case will be monitored and checked by ANSYS Fluent as a means to determine the convergence of the solution. It is a good practice to also create and plot a surface report definition that can help evaluate whether the solution is truly converged. You will do this in the next step.

3. Create a surface report definition of average temperature at the outlet (pressure-outlet-7).

Solution  $\rightarrow$  Reports  $\rightarrow$  Definitions  $\rightarrow$  New  $\rightarrow$  Surface Report  $\rightarrow$  Mass-Weighted Average...

Surface Report Definition	×
Name	Report Type
outlet-temp-avg	Mass-Weighted Average
Options Per Surface Average Over 1 Report Files [0/0] Report Plots [0/0]	Custom Vectors Vectors of Custom Vectors Field Variable Temperature Static Temperature Surfaces Filter Text default-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-5 wall
Create Create Report File Report Plot Frequency 3 Print to Console Create Output Parameter	Highlight Surfaces
	OK Compute Cancel Help

### Note:

You can also access the **Surface Report Definition** dialog box by right-clicking **Report Definitions** in the tree (under **Solution**) and selecting **New/Surface Report/Mass-Weighted Average...** from the menu that opens.

- a. Enter outlet-temp-avg for the Name of the report definition.
- b. Enable **Report File**, **Report Plot**, and **Print to Console** in the **Create** group box.

During a solution run, ANSYS Fluent will write solution convergence data in a report file, plot the solution convergence history in a graphics window, and print the value of the report definition to the console.

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

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

- d. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- e. Select **pressure-outlet-7** from the **Surfaces** selection list.
- f. Click **OK** to save the surface report definition and close the **Surface Report Definition** dialog box.

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

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

Edit Report File			×	
Name outlet-temp-avg-rfile				
Available Report Definitions [0/0]		Selected Report Definitions [0/1]	<b>-</b>	
		outlet-temp-avg		
	Add>>			
	< <remove< td=""><td></td><td></td></remove<>			
File Name		New - Edit		
\outlet-temp-avg-rfile.out Browse				
Full File Name				
Get Data Every 3 🌲 iteration 💌				
✓ Print to Console				
OK Cancel Help				

The dialog box is automatically populated with data from the **outlet-temp-avg** report definition.

a. Verify that outlet-temp-avg is in the Selected Report Definitions list.

If you had created multiple report definitions, the additional ones would be listed under Available **Report Definitions**, and you could use the Add>> and <<**Remove** buttons to manage which were written in this particular report definition file.

- b. (optional) Edit the name and location of the resulting file as necessary using the **File Name** field or **Browse...** button.
- c. Click **OK** to close the **Edit Report File** dialog box.
- 5. Create a convergence condition for **outlet-temp-avg**.

Active	Conditions	Report Definition	Stop Criterion	Ignore Iterations Before	Use Iterations	Print	Delete
✓	con-outlet-temp-avg	outlet-temp-	1e-5	20	15 🗘	✓	
Add							
	Condition	Every Iteration					
Choose	Condition	<b>A</b>	)				

**Solution**  $\rightarrow$  Reports  $\rightarrow$  Convergence...

- a. Click the **Add** button.
- b. Enter con-outlet-temp-avg for Conditions.
- c. Select outlet-temp-avg from the Report Definition drop-down list.
- d. Enter 1e-5 for Stop Criterion.
- e. Enter 20 for Ignore Iterations Before.
- f. Enter 15 for Use Iterations.
- g. Enable Print.

- h. Set **Every Iteration** to 3.
- i. Click **OK** to save the convergence condition settings and close the **Convergence Conditions** dialog box.

These settings will cause Fluent to consider the solution converged when the surface report definition value for each of the previous 15 iterations is within 0.001% of the current value. Convergence of the values will be checked every 3 iterations. The first 20 iterations will be ignored, allowing for any initial solution dynamics to settle out. Note that the value printed to the console is the deviation between the current and previous iteration values only.

6. Initialize the flow field using the **Initialization** group box of the **Solution** ribbon tab.

Solution → Initialization						
Initialization						
Method	Patch					
Hybrid More Settings Standard Options	🖒 Reset Statistics	t=0				
	🗇 Reset DPM	Initialize				

- a. Retain the default selection of Hybrid from the Method list.
- b. Click Initialize.
- 7. Save the case file (elbow1.cas.h5).

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

📕 Select File							?	$\times$
Look in: C:\Tutorials\introduction					- G	0 0	ß	:
S My Comput Documents	er Name	Size h	Туре	Date Modified		_		
	w.cas.h5							ок
Files of type: All	Case Files (*.cas	* *.msh* *.MS	SH* )				Ψ	Cancel
Filter String								Filter
✓ Display Mesh A	After Reading							

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

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

- b. Enter elbow1.cas.h5 for Case File.
- c. Ensure that the default **Write Binary Files** option is enabled, so that a binary file will be written.
- d. Click **OK** to save the case file and close the **Select File** dialog box.
- 8. Start the calculation by requesting 150 iterations in the **Solution** ribbon tab (**Run Calculation** group box).

### Solution → Run Calculation

Run Calculation					
🗹 Input Summary	📌 Run Calculation				
✓ Check Case	Time Scale Factor 1	•	$\bigcirc$		
♦ Update Dynamic Mesh	No. of Iterations 150		Calculate		

- a. Enter 150 for **No. of Iterations**.
- b. Click Calculate.

#### Note:

By starting the calculation, you are also starting to save the surface report data at the rate specified in the **Surface Report Definition** dialog box. If a file already exists in your working directory with the name you specified in the **Edit Report File** dialog box, then a **Question** dialog box will open, asking if you would like to append the new data to the existing file. Click **No** in the **Question** dialog box, and then click **OK** in the **Warning** dialog box that follows to overwrite the existing file.

As the calculation progresses, the surface report history will be plotted in the **outlet-temp-avg-rplot** tab in the graphics window (Figure 1.3: Convergence History of the Mass-Weighted Average Temperature (p. 37)).

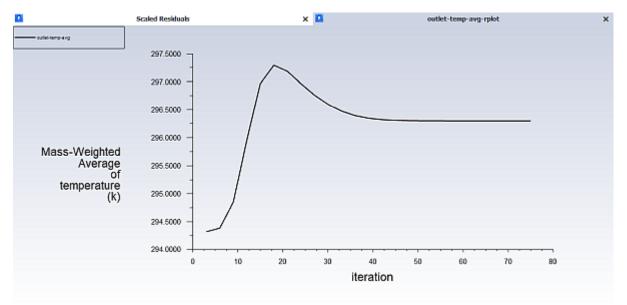
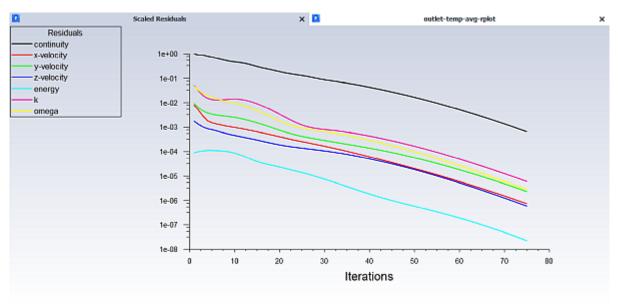


Figure 1.3: Convergence History of the Mass-Weighted Average Temperature

Similarly, the residuals history will be plotted in the **Scaled Residuals** tab in the graphics window (Figure 1.4: Residuals (p. 38)).



#### Figure 1.4: Residuals

#### Note:

You can monitor the two convergence plots simultaneously by right-clicking a tab in the graphics window and selecting **SubWindow View** from the menu that opens. To return to a tabbed graphics window view, right-click a graphics window title area and select **Tabbed View**.

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

The solution will be stopped by ANSYS Fluent when any of the following occur:

- the surface report definition converges to within the tolerance specified in the **Convergence Conditions** dialog box
- the residual monitors converge to within the tolerances specified in the **Residual Monitors** dialog box
- the number of iterations you requested in the Run Calculation task page has been reached

In this case, the solution is stopped when the convergence criterion on outlet temperature is satisfied. The exact number of iterations for convergence will vary, depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

9. Examine the plots for convergence (Figure 1.3: Convergence History of the Mass-Weighted Average Temperature (p. 37) and Figure 1.4: Residuals (p. 38)).

#### Note:

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

There are three indicators that convergence has been reached:

• The residuals have decreased to a sufficient degree.

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

• The solution no longer changes with more iterations.

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

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

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

10. Examine the mass flux report for convergence using the **Results** ribbon tab.



Flux Reports		×
Options		
Mass Flow Rate	Boundaries Filter Text 🔂 🛃 🛼	Results
O Total Heat Transfer Rate	default-interior	
Radiation Heat Transfer Rate	pressure-outlet-7 symmetry	-1.915872265440582
	velocity-inlet-5	1.614609710153648
	velocity-inlet-6	0.3012634374729083
	wall	
	*	<
Save Output Parameter		Net Results (kg/s) 8.82186e-07
	Compute Write Close Help	•

- a. Ensure that Mass Flow Rate is selected from the Options list.
- b. Select pressure-outlet-7, velocity-inlet-5, and velocity-inlet-6 from the Boundaries selection list.
- c. Click Compute.

The individual and net results of the computation will be displayed in the **Results** and **Net Results** boxes, respectively, in the **Flux Reports** dialog box, as well as in the console.

The sum of the flux for the inlets should be very close to the sum of the flux for the outlets. The net results show that the imbalance in this case is well below the 0.2% criterion suggested previously.

- d. Close the Flux Reports dialog box.
- 11. Save the data file (elbow1.dat.h5).



In later steps of this tutorial you will save additional case and data files with different suffixes.

## 1.4.7. Displaying the Preliminary Solution

In the steps that follow, you will visualize various aspects of the flow for the preliminary solution using the **Results** ribbon tab.

1. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.5: Predicted Velocity Distribution after the Initial Calculation (p. 43)).

Results  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

Contours	×
Contour Name	
contour-vel	
Options	Contours of
✓ Filled	Velocity
✓ Node Values	Velocity Magnitude
Boundary Values	Min (m/s) Max (m/s)
✓ Contour Lines ✓ Global Range	0 1.37504
Auto Range Auto Range Clip to Range Draw Profiles Draw Mesh Coloring Banded	Surfaces Filter Text pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall
Smooth	
Colormap Options	New Surface 🚽
s	ave/Display Compute Close Help

- a. Enter contour-vel for Contour Name.
- b. Enable **Filled** in the **Options** group box.
- c. Ensure that **Node Values** and **Boundary Values** are enabled in the **Options** group box.
- d. Select **Banded** in the **Coloring** group box.
- e. Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.
- f. Select **symmetry** from the **Surfaces** selection list.

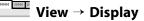
g. Click Save/Display to display the contours in the active graphics window. Clicking the Fit to Window

icon ( will cause the object to fit exactly and be centered in the window.

Note:

If you cannot see the velocity contour display, select the appropriate tab in the graphics window.

h. Close the **Contours** dialog box.



<u>File</u>	Domain	Physics	User-Defin	ed Solutio	n	Results		View	Parall	lel
	Display		Grap	phics			Mouse			
🖭 Views	✓ Headlight	✓ Lighting	Q Lights	💋 Compose	Left	mouse-rotate	▼ Prol	e short des	cription 👻	
E Options		🌠 Gouraud գ	Colormap		Middle	mouse-zoom	*	Fluent Def	aults	
Camera			🜱 Annotate		Right	mouse-probe	*	Workbench D	Defaults	

Disable the Headlight and Lighting options.

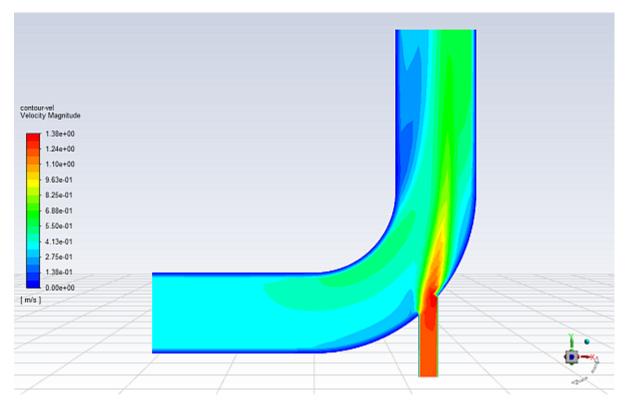


Figure 1.5: Predicted Velocity Distribution after the Initial Calculation

#### Extra:

When you probe a point in the displayed domain with the right mouse button or the probe tool, the level of the corresponding contour is highlighted in the colormap in the graphics window, and is also reported in the console.

2. Create and display a definition for temperature contours on the symmetry plane (Figure 1.6: Predicted Temperature Distribution after the Initial Calculation (p. 45)).

### Results $\rightarrow$ Graphics $\rightarrow$ Contours $\rightarrow$ New...

You can create contour definitions and save them for later use.

Contours	×
Contour Name	
contour-temp	
Options	Contours of
✓ Filled	Temperature 💌
Node Values	Static Temperature
Boundary Values     Contour Lines	Min (k) Max (k)
✓ Global Range	293.1471 313.1574
Auto Range Clip to Range Draw Profiles Draw Mesh Coloring Banded Smooth	Surfaces Filter Text pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall
Colormap Options	New Surface 🚽
s	ave/Display Compute Close Help

- a. Enter contour-temp for Contour Name.
- b. Select Temperature... and Static Temperature from the Contours of drop-down lists.
- c. Select **symmetry** from the **Surfaces** selection list.
- d. Click Save/Display and close the Contours dialog box.

The new **contour-temp** definition appears under the **Results/Graphics/Contours** tree branch. To edit your contour definition, right-click it and select **Edit...** from the menu that opens.

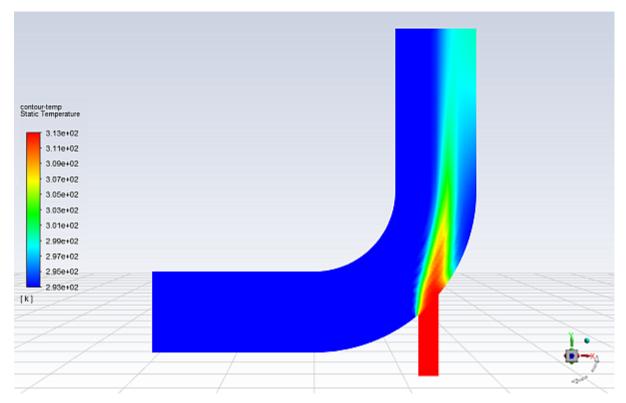


Figure 1.6: Predicted Temperature Distribution after the Initial Calculation

3. Display velocity vectors on the symmetry plane (Figure 1.9: Magnified View of Resized Velocity Vectors (p. 49)).



Vectors	×
Vector Name	
vector-vel	
Options	Vectors of
✓ Global Range	Velocity 💌
Auto Range	Color by
Clip to Range	Velocity 💌
<ul> <li>Auto Scale</li> <li>Draw Mesh</li> </ul>	Velocity Magnitude
Style	Min (m/s) Max (m/s)
3d arrow 💌	0.1344519 1.404483
Scale Skip	Surfaces Filter Text
Vector Options	symmetry
Custom Vectors	velocity-inlet-5 velocity-inlet-6 wall
Colormap Options	
	New Surface 🚽
s	ave/Display Compute Close Help

- a. Enter vector-vel for Vector Name.
- b. Select **arrow** from the **Style** drop-down box.
- c. Select **symmetry** from the **Surfaces** selection list.
- d. Click **Save/Display** to plot the velocity vectors.

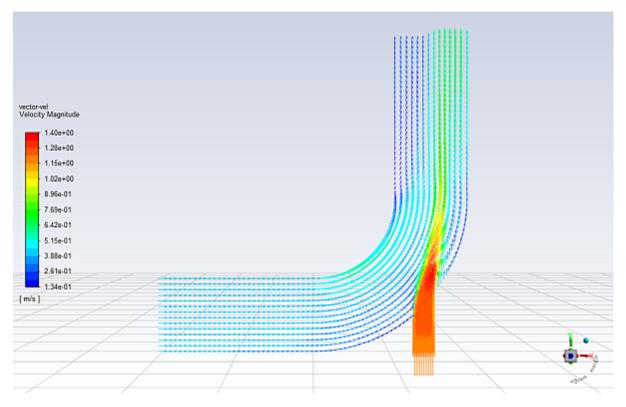


Figure 1.7: Velocity Vectors Colored by Velocity Magnitude

The **Auto Scale** option is enabled by default in the **Options** group box. This scaling sometimes creates vectors that are too small or too large in the majority of the domain. You can improve the clarity by adjusting the **Scale** and **Skip** settings, thereby changing the size and number of the vectors when they are displayed.

- e. Enter 4 for Scale.
- f. Set Skip to 2.
- g. Click Save/Display again to redisplay the vectors.

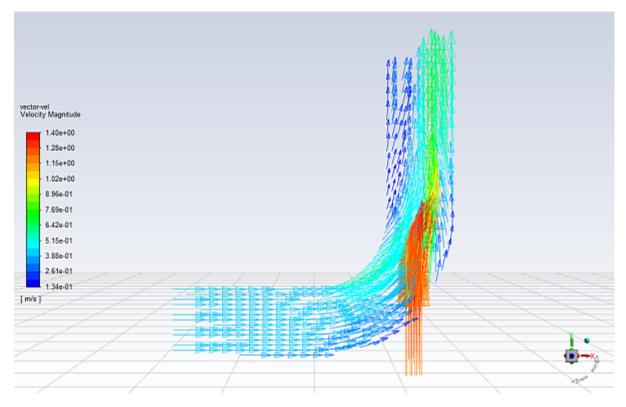


Figure 1.8: Resized Velocity Vectors

- h. Close the **Vectors** dialog box.
- i. Zoom in on the vectors in the display.

To manipulate the image, refer to Table 1.1: View Manipulation Instructions (p. 8). The image will be redisplayed at a higher magnification (Figure 1.9: Magnified View of Resized Velocity Vectors (p. 49)).

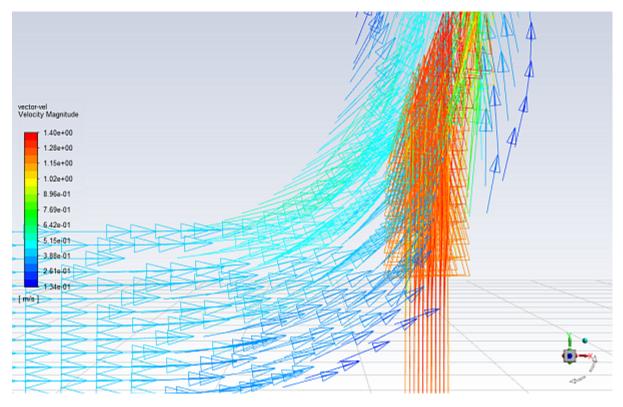


Figure 1.9: Magnified View of Resized Velocity Vectors

j. Zoom out to the original view.

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

4. Create a line at the centerline of the outlet. For this task, you will use the **Surface** group box of the **Results** tab.



**Results**  $\rightarrow$  Surface  $\rightarrow$  Create  $\rightarrow$  Iso-Surface...

Iso-Surface			×
New Surface Name z=0_outlet		From Surface Filter Text	- <b>=</b> , <b>=</b> , <b>=</b> ,
Surface of Constant Mesh Z-Coordinate Min (in) 0 Iso-Values (in) 0	• Max (in) 2	default-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall From Zones Filter Text fluid	
	Create	ompute Close Help	

- a. Enter z=0\_outlet for New Surface Name.
- b. Select Mesh... and Z-Coordinate from the Surface of Constant drop-down lists.
- c. Click **Compute** to obtain the extent of the mesh in the z-direction.

The range of values in the z-direction is displayed in the **Min** and **Max** fields.

- d. Retain the default value of 0 inches for Iso-Values.
- e. Select pressure-outlet-7 from the From Surface selection list.
- f. Click Create.

The new line surface representing the intersection of the plane z=0 and the surface pressureoutlet-7 is created, and its name **z=0\_outlet** appears in the **From Surface** selection list.

#### Note:

- After the line surface **z=0\_outlet** is created, a new entry will automatically be generated for **New Surface Name**, in case you would like to create another surface.
- If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the **Surfaces** dialog box.
- g. Close the **Iso-Surface** dialog box.

5. Display and save an XY plot of the temperature profile across the centerline of the outlet for the initial solution (Figure 1.10: Outlet Temperature Profile for the Initial Solution (p. 52)).

Solution XY Plot		×
XY Plot Name		
xy-outlet-temp		
Options	Plot Direction Y Axis Function	
✓ Node Values	X 1 Temperature	-
Position on X Axis     Position on Y Axis	Y 0 Z 0 Static Temperature	•
Write to File	X Axis Function	
Order Points	Direction Vector	-
File Data [0/0]	Image: Surface Sector Symmetry         Image: Free Data         Surface Sector Symmetry         Image: Surface Sector Sec	
s	ave/Plot Axes Curves Close Help	

Results  $\rightarrow$  Plots  $\rightarrow$  XY Plot  $\rightarrow$  New...

- a. Enter xy-outlet-temp for XY Plot Name.
- b. Select Temperature... and Static Temperature from the Y Axis Function drop-down lists.
- c. Select the **z=0\_outlet** surface you just created from the **Surfaces** selection list.
- d. Click Save/Plot.
- e. Enable Write to File in the Options group box.

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

- f. Click Write....
  - i. In the Select File dialog box, enter outlet\_temp1.xy for XY File.
  - ii. Click **OK** to save the temperature data and close the **Select File** dialog box.
- g. Close the Solution XY Plot dialog box.

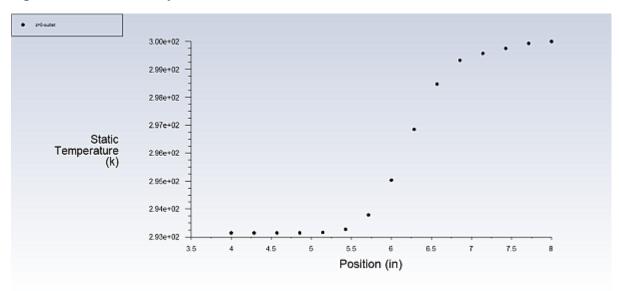


Figure 1.10: Outlet Temperature Profile for the Initial Solution

6. Define a custom field function for the dynamic head formula  $(\rho \cdot |V|^2/2)$ .

User-Defined  $\rightarrow$  Field Functions  $\rightarrow$  Custom...

Custom Field Function Calculator	$\times$		
Definition			
density *  V  ^ 2 / 2			
+       -       X       / y^x ABS       Select Operand Field Functions from         INV       sin       cos       tan       In       log10         0       1       2       3       4       SQRT         5       6       7       8       9       CE/C         (       )       PI       e       DEL			
New Function Name dynamic-head			
Define Manage Close Help			

- a. Select **Density...** and **Density** from the **Field Functions** drop-down lists, and click the **Select** button to add **density** to the **Definition** field.
- b. Click the **X** button to add the multiplication symbol to the **Definition** field.
- c. Select Velocity... and Velocity Magnitude from the Field Functions drop-down lists, and click the Select button to add |V| to the Definition field.

- d. Click **y**^**x** to raise the last entry in the **Definition** field to a power, and click **2** for the power.
- e. Click the / button to add the division symbol to the **Definition** field, and then click **2**.
- f. Enter dynamic-head for New Function Name.
- g. Click Define and close the Custom Field Function Calculator dialog box.

The **dynamic-head** tree item will appear under the **Parameters & Customization/Custom Field Functions** tree branch.

7. Display filled contours of the custom field function (Figure 1.11: Contours of the Dynamic Head Custom Field Function (p. 54)).

**Results**  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  New...

Contours	×
Contour Name contour-dynamic-head	
Options	Contours of
✓ Filled	Custom Field Functions
✓ Node Values	dynamic-head
Boundary Values	Min Max
Contour Lines	0 0
✓ Global Range	
✓ Auto Range	Surfaces Filter Text 🗧 🗧 🔫
Clip to Range	
Draw Profiles	pressure-outlet-7 symmetry
Draw Mesh	velocity-inlet-5
Coloring Banded Smooth	velocity-inlet-6 wall z=0_outlet
Colormap Options	
	New Surface 🚽
s	ave/Display Compute Close Help

a. Enter contour-dynamic-head for Contour Name.

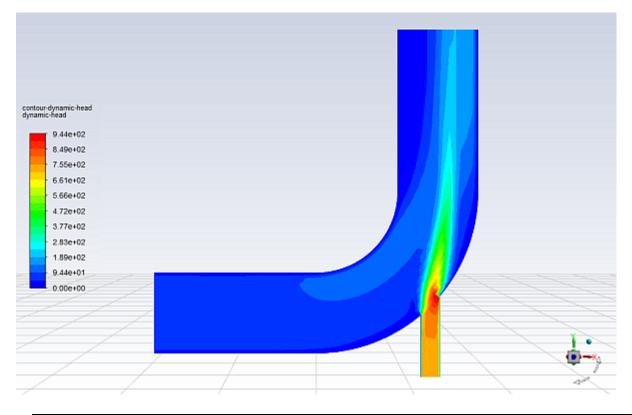
- b. Select **Banded** in the **Coloring** group box.
- c. Select Custom Field Functions... and dynamic-head from the Contours of drop-down lists.

Tip:

**Custom Field Functions...** is at the top of the upper **Contours of** drop-down list.

- d. Select symmetry from the Surfaces selection list.
- e. Click Save/Display and close the Contours dialog box.

Figure 1.11: Contours of the Dynamic Head Custom Field Function



#### Note:

You may need to change the view by zooming out after the last vector display, if you have not already done so.

8. Save the settings for the custom field function by writing the case and data files (elbow1.cas.h5 and elbow1.dat.h5).

# **File** $\rightarrow$ Write $\rightarrow$ Case & Data...

a. Ensure that elbow1.cas.h5 is entered for Case/Data File.

Note:

When you write the case and data file at the same time, it does not matter whether you specify the file name with a .cas.h5 or .dat.h5 extension, as both will be saved.

- b. Click **OK** to save the files and close the **Select File** dialog box.
- c. Click **OK** to overwrite the files that you had saved earlier.

## 1.4.8. Adapting the Mesh

For the first run of this tutorial, you have solved the elbow problem using a fairly coarse mesh. The elbow solution can be improved further by refining the mesh to better resolve the flow details. ANSYS Fluent provides a built-in capability to easily adapt (locally refine) the mesh according to solution gradients. In the following steps you will adapt the mesh based on the temperature gradients in the current solution and compare the results with the previous results.

1. Define **Cell Registers** to Adapt the mesh in the regions of high temperature gradient.

**Solution**  $\rightarrow$  Cell Registers  $\stackrel{\frown}{\longrightarrow}$  New  $\rightarrow$  Field Variable...

Filter Text		General		?
<ul> <li>Setup</li> <li>General</li> <li>Models</li> <li>Materials</li> <li>Cell Zone Condition</li> <li>Boundary Condition</li> <li>Dynamic Mesh</li> <li>Reference Values</li> <li>Keference Frames</li> </ul>	ons		Velocity Formulation	
<ul> <li>Mamed Expression</li> <li>Solution</li> <li>Methods</li> <li>Controls</li> </ul>	ns	<ul> <li>Pressure-Based</li> <li>Density-Based</li> <li>Time</li> </ul>	Absolute     Relative	
<ul> <li>Report Definitions</li> <li>Monitors</li> <li>Cell Registers</li> <li>Initialization</li> </ul>	Edit	<ul> <li>Steady</li> <li>Transient</li> </ul>		
<ul> <li>Calculation Activitie</li> <li>Run Calculation</li> <li>Results</li> <li>Surfaces</li> <li>Graphics</li> <li>Plots</li> <li>Scene</li> <li>Animations</li> <li>Reports</li> <li>Parameters &amp; Customi</li> </ul>	New Expand All Collapse All	Region Boundary Field Variable Limiter Residuals Volume Yplus/Ystar		
🐨 Parameters & Customi	zation	rpius/rstar		

Name_curvature_0			
Туре	Curvature of		
Cells More Than	Temperature		-
Derivative Option	Static Temperature		-
Curvature •	Min	Max	
Scaling Option	4.930381e-32	0.03171117	]
None 💌	Cells having value mo	re than 0.003	
Save/Display Sa	Display Options	Compute Close Help	

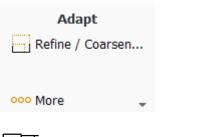
- a. Select Cells More Than from the Type drop-down list.
- b. Select Curvature from the Derivative Option drop-down list.
- c. Select **Temperature...** and **Static Temperature** from the **Curvature of** drop-down list.
- d. Click **Compute**.

ANSYS Fluent will update the **Min** and **Max** values to show the minimum and maximum temperature gradient.

e. Enter a value of 0.003 for the **Cells having value more than**.

A general rule is to use 10% of the maximum gradient when setting the value for refinement.

- f. Click Save and close the Field Variable Register daialog box.
- 2. Setup mesh adaption using the **Cell Registers**. For this task, you will use the **Adapt** group box in the **Domain** ribbon tab.



<sup>™</sup> Domain → Adapt → Refine / Coarsen...

Adaption Controls	×	
Refinement Criterion curvature_0	•	
Coarsening Criterion	•	
Maximu	ım Refinement Level 2	
Minim	um Cell Volume (m3) 0	
Dynamic Adaption Predefined Criter		
	Cell Registers 💌	
Advanced Controls		
Display Options		
OK Adapt Display Cancel Help		

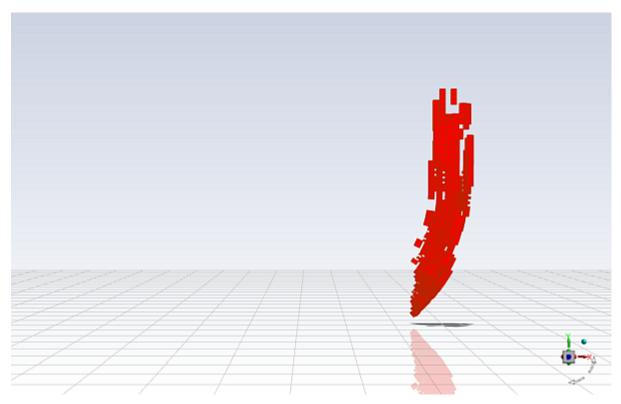
a. Select the previously defined **curvature\_0** cell register from the **Refinement Criterion** drop-down list.

ANSYS Fluent will not coarsen beyond the original mesh for a 3D mesh. Hence, it is not necessary to select the **Coarsening Criterion** in this instance.

- b. Click **Adapt**.
- c. Click **Display**.

ANSYS Fluent will display the cells marked for adaption in the graphics window (Figure 1.12: Cells Marked for Adaption (p. 59)).

Figure 1.12: Cells Marked for Adaption



**Extra** You can change the way ANSYS Fluent displays cells marked for adaption (Figure 1.13: Alternative Display of Cells Marked for Adaption (p. 61)) by performing the following steps:

i. Click **Display Options...** in the **Adaption Controls** dialog box to open the **Display Options** - **Adaption** dialog box.

Options	Refinement Cells	Coarsening Cells	Common Cells
✓ Draw Mesh	✓ Faces	✓ Faces	✓ Faces
	✓ Edges	✓ Edges	✓ Edges
	Centroid	Centroid	Centroid
	Face Color red	Tace Color green	Face Color yellow
	Edge Color tan	▼ Edge Color cyan	▼ Edge Color dark gray ▼
	Edge Width 1	C Edge Width 1	Edge Width 1

ii. Enable **Draw Mesh** in the **Options** group box.

📕 Mesh Display		×
Options Nodes Sedges Faces Partitions Overset	Edge Type All Feature Outline	Surfaces Filter Text Gefault-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall
Shrink Factor Fe		z=0_outlet
Adjacency	Dis	New Surface _ play Colors Close Help

- iii. Ensure that only the **Edges** option is enabled in the **Options** group box.
- iv. Select Feature from the Edge Type list.
- v. Select all of the items except **z=0\_outlet** from the **Surfaces** selection list.
- vi. Click **Display** and close the **Mesh Display** dialog box.
- vii. Click OK to close the Display Options Adaption dialog box.
- viii.Click **Display** in the **Adaption Controls** dialog box.
- ix. Rotate the view and zoom in to get the display shown in Figure 1.13: Alternative Display of Cells Marked for Adaption (p. 61).

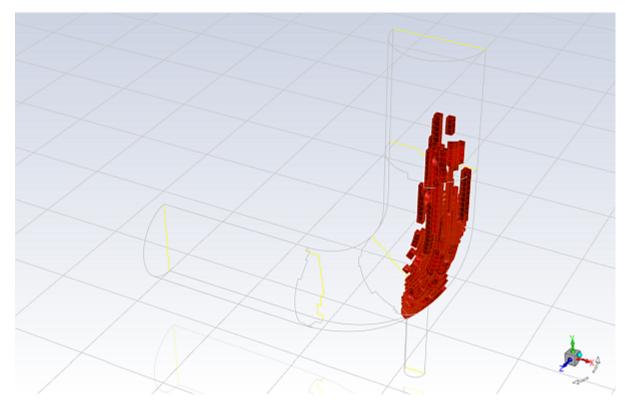


Figure 1.13: Alternative Display of Cells Marked for Adaption

- x. After viewing the marked cells, rotate the view back and zoom out again.
- xi. Click **OK** to close the **Adaption Controls** dialog box.
- 3. Display the adapted mesh (Figure 1.14: The Adapted Mesh (p. 63)).

**■** Domain → Mesh → Display...

🚺 Mesh Display	y	×				
Options	Edge Type					
Nodes	All	Surfaces Filter Text 🗾 🔂 🗮 🗫				
✓ Edges	○ Feature	pressure-outlet-7				
Faces	Outline	symmetry				
Partitions		velocity-inlet-5				
Overset		velocity-inlet-6 wall				
Shrink Factor F	eature Angle	z=0_outlet				
Adjacency		New Surface 🚽				
Display Colors Close Help						

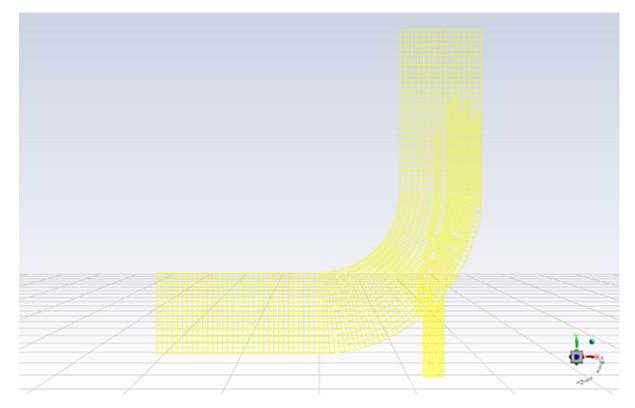
- a. Select All from the Edge Type list.
- b. Deselect all of the highlighted items from the **Surfaces** selection list except for **symmetry**.

#### Tip:

To deselect all surfaces, click the **Deselect All Shown** button () at the top of the **Surfaces** selection list. Then select the desired surface from the **Surfaces** selection list.

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

#### Figure 1.14: The Adapted Mesh



4. Request an additional 90 iterations.

Solution $\rightarrow$ Run Calculation $\rightarrow$ Calculate									
	<b>Run Calculation</b>								
🗹 Input Summary	🃌 Run Calculation								
🗹 Check Case	Time Scale Factor 1		-	$\bigcirc$					
✔ Update Dynamic Mesh	No. of Iterations 90	\$		Calculate					

The solution will converge as shown in Figure 1.15: The Complete Residual History (p. 64) and Figure 1.16: Convergence History of Mass-Weighted Average Temperature (p. 64).

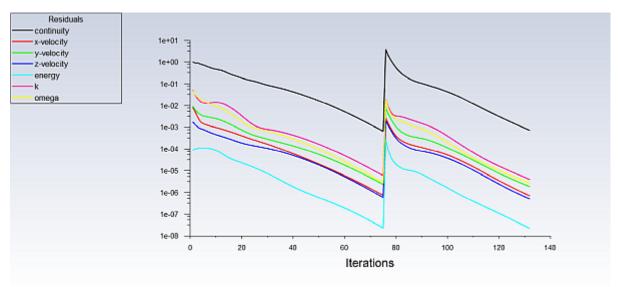
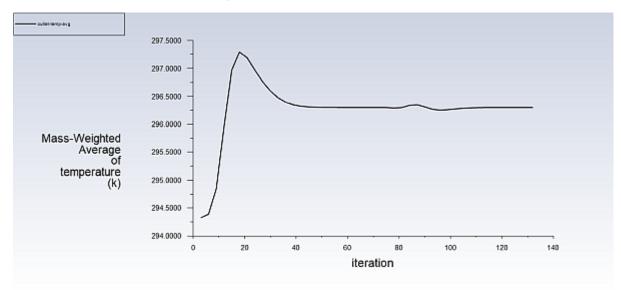


Figure 1.15: The Complete Residual History

Figure 1.16: Convergence History of Mass-Weighted Average Temperature



5. Save the case and data files for the Coupled solver solution with an adapted mesh (elbow2.cas.h5 and elbow2.dat.h5).

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

- a. Enter elbow2.h5 for Case/Data File.
- b. Click **OK** to save the files and close the **Select File** dialog box.

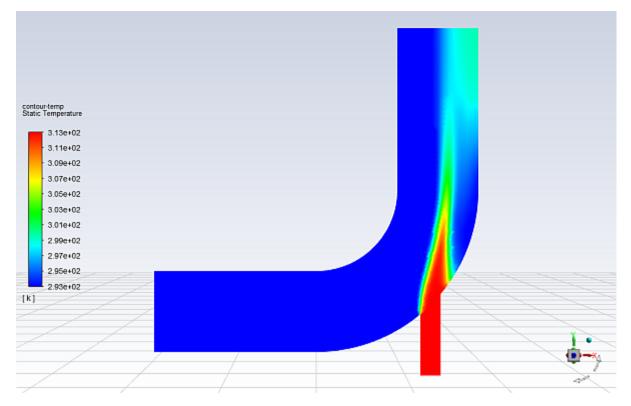
The files elbow2.cas.h5 and elbow2.dat.h5 will be saved in your default folder.

6. Display the temperature distribution (using node values) on the revised mesh using the temperature contours definition that you created earlier (Figure 1.17: Filled Contours of Temperature Using the Adapted Mesh (p. 65)).

Right-click the **Results/Graphics/Contours/contour-temp** tree item and select **Display** from the menu that opens.

**Results**  $\rightarrow$  Graphics  $\rightarrow$  Contours  $\rightarrow$  contour-temp  $\stackrel{\text{D}}{\rightarrow}$  Display





7. Display and save an XY plot of the temperature profile across the centerline of the outlet for the adapted solution (Figure 1.18: Outlet Temperature Profile for the Adapted Coupled Solver Solution (p. 66)).



a. Click Save/Plot to display the XY plot.

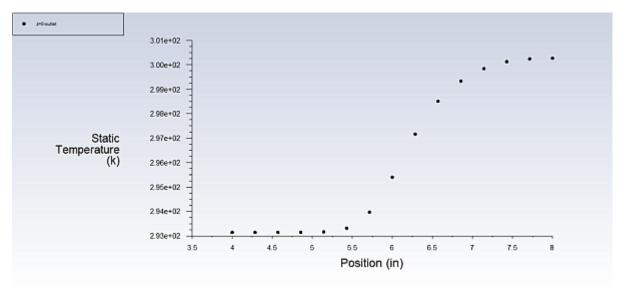


Figure 1.18: Outlet Temperature Profile for the Adapted Coupled Solver Solution

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

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

- c. Click Write....
  - i. In the Select File dialog box, enter outlet\_temp2.xy for XY File.
  - ii. Click **OK** to save the temperature data.
- d. Close the **Solution XY Plot** dialog box.
- 8. Display the outlet temperature profiles for both solutions on a single plot (Figure 1.19: Outlet Temperature Profiles for the Two Solutions (p. 68)).
  - a. Open the Plot Data Sources dialog box.

**Results**  $\rightarrow$  Plots  $\rightarrow$  Data Sources...

Plot Data Sources			×
Data Sources		Variables	
Load File     outlet_temp1.xy       Add XY Plot     outlet_temp2.xy       Add Report Plot     Remove Data Source	Data Sets (1/1)	X Axis Variable Position Static Temperati Static Temperati	
C:/Tutorials/introduction/outlet	_temp2.xy		
Curves		Plo	t
Curve Information [1/2]	Before Adaption	Remove	Title Static Temperature gend Label Static Temperature (Axis Label Position Axis Label Static Temperature
Change Legend Entr	Y Adapted Mesh		
	Plot Axes Curves Close	Help	

b. Click the **Load File...** button to open the **Select File** dialog box.

📧 Select File										?		$\times$
Look in:	C:\	\Tutorials\in	troduction			•	G	0	0	ß	::	Ξ
S My Con	nputer	Name	- S	ize	Туре	Date Modified						
	-	outlet_t	emp1.xy	367 bytes	xy File	11/13/2018 9:27 AM						
	ents	outlet_t	emp2.xy	361 bytes	xy File	11/13/2018 11:20 AM	И					
2												
XY File	outlet_t	temp2.xy									01	٢)
Files of type:	XY File	es (*.xy *.ou	ıt)							• (	Can	cel
Filter String										$\Box$	Filt	er
										R	emo	ve
C:/Tutorials/i C:/Tutorials/i												

i. Select outlet\_temp1.xy and outlet\_temp2.xy.

Each of these files will be listed with their folder path in the bottom list to indicate that they have been selected.

#### Tip:

If you select a file by mistake, simply click the file in the bottom list and then click **Remove**.

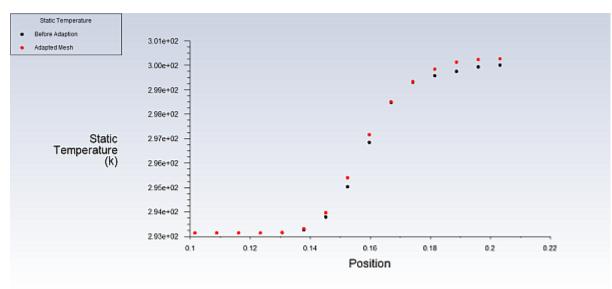
- ii. Click **OK** to save the files and close the **Select File** dialog box.
- c. Select the folder path ending in **outlet\_temp1.xy** from the **Curve Information** selection list (**Curves** group box).
- d. Enter Before Adaption in the lower-right text-entry box.
- e. Click the Change Legend Entry button.

The item in the **Legend Entries** list for **outlet\_temp1.xy** will be changed to **Before Adaption**. This legend entry will be displayed in the upper-left corner of the XY plot generated in a later step.

- f. In a similar manner, change the legend entry for the folder path ending in **outlet\_temp2.xy** to be Adapted Mesh.
- g. Click **Plot** and close the **Plot Data Sources** dialog box.

Figure 1.19: Outlet Temperature Profiles for the Two Solutions (p. 68) shows the two temperature profiles at the centerline of the outlet. It is apparent by comparing both the shape of the profiles and the predicted outer wall temperature that the solution is highly dependent on the mesh and solution options. Specifically, further mesh adaption should be used in order to obtain a solution that is independent of the mesh.

Figure 1.19: Outlet Temperature Profiles for the Two Solutions



## 1.5. Summary

The solution results are changed by the adaption of the mesh, which indicates that a sufficiently refined mesh is required to obtain a mesh independent solution.

# Release 2020 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.