Динамика течения и теплообмен в смешивающихся потоках воздуха (Пример из официальной документации)

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.

1.3. Problem Description

The problem to be considered is shown schematically in <u>Figure 1.1: Problem Specification</u>. 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 <u>Typographical Conventions Used In This Manual</u> for detailed information.

1.4.1. Preparation

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

1.4.2. Launching ANSYS Fluent

• 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.

Meshing	Simulate a wide range of industrial applicatio and post-processing capabilities of ANSYS FI Get Started With	ons using the general-purpose s uent. Dimension	etup, solv
Solution	Case Case and Data) O 2D	
	Mesh Journal	0 3D	
	Recent Files	Options Double Precision Display Mesh After Readin Load ACT Start Server Parallel (Local Machine)	ng
		Solver Processes	4
		Solver GPGPUs per Machine	0

Ensure that the proper options are enabled.

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

Note: Fluent will retain your preferences for future sessions.

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

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

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

Meshing		Simulate a and post-p Get Starl	wide range processing ca ted With	of industrial applic apabilities of ANSYS	ations using the general-purpose : 5 Fluent. Dimension	setup, s	olve,
Solution		(Case	Case and Data) O 2D		
	18		tesh	Journal	─ ○ 3D		
cing		Recent F	ilos	AC	Options		
		Recent	lies		Double Precision		
					Display Mesh After Read	ing	
					Load ACT		
					Start Server		
					Parallel (Local Machine)		
					Solver Processes	4	1
~ Show Fewer	Options Y Sh	ow Learnin	ng Resource	25	Solver GPGPUs per Machine	0	3
~ Show Fewer General Options	Options Y Sh Parallel Settings	ow Learnin Remote	ng Resource Scheduler	25 Environment	Solver GPGPUs per Machine	0	
~ Show Fewer General Options	Options Y Sha Parallel Settings	ow Learnin Remote	ng Resource Scheduler	ès Environment	Solver GPGPUs per Machine	0	
~ Show Fewer General Options Pre/Post On Working Directo	Options V Shu Parallel Settings Iy IY	ow Learnin Remote	ng Resource Scheduler	25 Environment	Solver GPGPUs per Machine	0	
~ Show Fewer General Options Pre/Post On Working Directo	Options ✓ Sh Parallel Settings ly ry	ow Learnin Remote	ng Resource Scheduler	Environment	Solver GPGPUs per Machine	•	
Show Fewer General Options Pre/Post On Working Directo Fluent Root Path	Options ✓ Shu Parallel Settings ly ry	ow Learnin Remote	ng Resource Scheduler	ès Environment	Solver GPGPUs per Machine	•	
 Show Fewer General Options Pre/Post On Working Directo Fluent Root Path 	Options Y Sha Parallel Settings ly ry	ow Learnin Remote	ng Resource Scheduler	25 Environment	Solver GPGPUs per Machine	•	
~ Show Fewer General Options Pre/Post On Working Directo Fluent Root Path	Options ➤ Sha Parallel Settings ly ry	ow Learnin Remote	ng Resource Scheduler	25 Environment	Solver GPGPUs per Machine	•	;
~ Show Fewer General Options Pre/Post On Working Directo Fluent Root Path	Options Y Sha Parallel Settings ly ry	ow Learnin Remote	ng Resource Scheduler	es Environment	Solver GPGPUs per Machine	•	;
~ Show Fewer General Options Pre/Post On Working Directo Fluent Root Path	Options ✓ Shu Parallel Settings IY I	ow Learnin Remote	ng Resource Scheduler	25 Environment	Solver GPGPUs per Machine	•	
~ Show Fewer General Options Pre/Post On Working Directo Fluent Root Path	Options ✓ Sh Parallel Settings ly ry	ow Learnin	ng Resource Scheduler	25 Environment	Solver GPGPUs per Machine	•	

Click **OK** to launch ANSYS Fluent.

Data Data Data Data Bits Bits Bits Bits Diff Bits Bits Bits Diff Bits Bits Bits Bits Diff Bits Bits Bits Bits Bits Diff Bits	Contract Descent Gale Continue	New Network Description Operation Ope
Notike View Time Tell Software So	 Task Page Task Page Solo Desk 	Constructive Number Constructive
		V Shan ka pap a dabay Sama S

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

• 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.

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

My Computer Name Size Type Date M Documents elbow.msh 2.59 MiB msle 1M Image: Size of type: Mesh Files (*.msh* *.MSH*) OK	.ook in:	C:\	Tutorials\intro	Juction				*	9	9	0	Lo	E
Image: Second Secon	My Con	nouter	Name *	Size		Туре	Date M						
Mesh File OK Tiles of type: Mesh Files (*.msh* *.MSH*) * Cance	Docume	ents	elbow.msł	(<u> </u>	2.59 MiB	msle	1M						

- Select the mesh file by clicking **elbow.msh** in the **introduction** folder created when you unzipped the original file.
- Enable the Display Mesh After Reading in the Select File dialog box.
- 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.

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.

- 1. Click the z-axis.
- 2. Clicking the **Fit to Window** icon, , will cause the object to fit exactly and be centered in the window.
- 3. Figure 1.2: The Hexahedral Mesh for the Mixing Elbow



Figure 1.2: The Hexahedral Mesh for the Mixing Elbow

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.

$\textbf{Domain} \rightarrow \textbf{Mesh} \rightarrow \textbf{Check} \rightarrow \textbf{Perform Mesh Check}$

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

```
Domain Extents:
    x-coordinate: min (m) = -8.000000e+00, max (m) =
    8.000000e+00
    y-coordinate: min (m) = -9.134634e+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): 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.

📕 Scale N	/lesh				\times
Domain E	xtents			Scaling	
Xmin (in) Ymin (in) Zmin (in)	-8 -9.134634 0	Xmax (in) Ymax (in) Zmax (in)	8 8 2	Convert Units Specify Scaling Mesh Was Created Ir	Factors
View Lengt	th Unit In			Scaling Factors X 0.0254 Y 0.0254 Z 0.0254 Scale Unsc	cale
		Q	Close Help		

 $\textbf{Domain} \rightarrow \textbf{Mesh} \rightarrow \textbf{Scale...}$

Ensure that **Convert Units** is selected in the **Scaling** group box.

- 1. 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.
- 2. Click Scale to scale the mesh.
- 3. Warning: Be sure to click the Scale button only once.

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

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

Right click in the graphics window and select Refresh Display



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

1. Check the mesh.

$\mathbf{Domain} \rightarrow \mathbf{Mesh} \rightarrow \mathbf{Check} \rightarrow \mathbf{Perform} \; \mathbf{Mesh} \; \mathbf{Check}$

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.

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

 $\mathbf{Physics} \rightarrow \mathbf{Solver} \rightarrow \mathbf{General}$

eneral		(?)
lesh		
Scale C	heck Report Quality	
Display U	nits	
olver		
Гуре	Velocity Formulation	
• Pressure-Based	 Absolute 	
O Density-Based	O Relative	
íme		
Steady		
○ Transient		

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

	M	lodels	
	📶 Radiation	🕰 Multiphase	⊿ Structure
Energy	∦≓ Heat Exchanger	••• Species	Coustics
	Uiscous	🌽 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.

2. Enable heat transfer by activating the energy equation.

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

$\mathsf{Physics} \to \mathsf{Models} \to \mathsf{Energy}$

3. Enable the k- ω turbulence model.

 $\mathbf{Physics} \rightarrow \mathbf{Models} \rightarrow \mathbf{Viscous...}$

Model	Model Constants	
O Inviscid	Alpha*_inf	-
	1	
O Spalart-Allmaras (1 eqn)	Alpha_inf	
🔿 k-epsilon (2 eqn)	0.52	
k-omega (2 eqn)	Beta*_inf	
O Transition k-kl-omega (3 eqn)	0.09	
O Transition SST (4 eqn)	al	
O Reynolds Stress (7 eqn)	0.31	
 Scale-Adaptive Simulation (SAS) Detected Edde Simulation (SEC) 	Beta i (Inner)	
Detached Eddy Simulation (DES)	0.075	
	Beta i (Outer)	- 1
k-omega Model		
○ Standard	User-Defined Functions	
O GEKO	Turbulent Viscosity	
OBSL	none	Ŧ
• SST	Prandtl Numbers	
k-omega Options	Energy Prandtl Number	
Low-Re Corrections	none	*
Ontions	Wall Prandtl Number	
Viscous Heating	none	*
Curvature Correction		-
Production Kato-Launder		
✓ Production Limiter		

- 4. Retain the default selection of **k-omega** from the **Model** list.
- 5. Retain the default selection of **SST** in the **k-omega Model** group box.
- 6. Click **OK** to accept all the other default settings and close the **Viscous Model** dialog box.
- 7. 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.

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.

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

Name	Mat	erial Type	order materials by
water-liquid) (flui	d	Name
Chemical Formula	Flue	ent Fluid Materials	Chemical Formula
h2o <l></l>	wa	ter-liquid (h2o <l>)</l>	• Fluent Database
Mi		ture	tteor Dofined Database
	no	ne	* Oser-Denned Database.
	Properties		
	Density (kg/m3)	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

 $\mathbf{Physics} \rightarrow \mathbf{Materials} \rightarrow \mathbf{Create/Edit...}$

Click the **Fluent Database...** button to access pre-defined materials.

Select **water-liquid (h2o < l >)** from the materials list and click **Copy**, then close the **Fluent Database...** dialog box.

Fluent Fluid Materials [1/563]		
vinyl-trichlorosilane (sicl3ch2ch)	Order Materials by	1
water-liquid (h2o <l>)</l>	Name	
water-vapor (h2o) wood-volatiles (wood_vol)	Chemical Form	ıla
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	

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 **Materials** tree branch.

Close the Create/Edit Materials dialog box.

Set up the cell zone conditions for the fluid zone (**fluid**) using the **Zones** group box of the **Physics** ribbon tab.



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

$\mathbf{Physics} \rightarrow \mathbf{Zones} \rightarrow \mathbf{Cell} \ \mathbf{Zones}$

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

Cell Zone Conditions	?
Zone Filter Text	=
fluid	
Phase Type ID	
Edit Copy Profiles	
Parameters Operating Conditions Display Mesh Operating Conditions	
Porous Formulation	
O Physical Velocity	

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

e En	nbedded LES	Reaction	Source Terms	Fixed Values	Multiphase
	Rotation-	-Axis Direc	tion		
•	X 0				-
	YO				
	Z 1				-
	e Er	e Embedded LES Rotation • X 0 • Y 0 • Z 1	e Embedded LES Reaction Rotation-Axis Direc X 0 Y 0 Z 1	e Embedded LES Reaction Source Terms Rotation-Axis Direction * X 0 * Y 0 * Z 1	e Embedded LES Reaction Source Terms Fixed Values Rotation-Axis Direction Y 0 Y 0 Z 1

Select water-liquid from the Material Name drop-down list.

Click **OK** to close the **Fluid** dialog box.

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.

	Zones
	Cell Zones
	Boundaries
	Profiles
• /-	

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 Condition	ons	?
Zone Filter Text		F
 Inlet velocity-inlet velocity-inlet Internal default-inter Outlet pressure-ou Symmetry symmetry Wall wall 	t-5 t-6 rior ıtlet-7	
Phase Type mixture	ID • 1	
Edit	Copy	
Display Mesh	Operating Conditions	
	Periodic Conditions	
Highlight Zone		

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.

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

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

Velocity l	nlet						×	
velocity-inlet	:-5							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS	
	Velocity Spe	ecification Me	thod Magnit	ude, Norma	l to Boundary		•	
Reference Frame Absolute								
Velocity Magnitude (m/s) 0.4								
Supersonic/	Initial Gauge	Pressure (pa	iscal) 0				•	
	Turbule	ence						
	Spec	ification Met	hod Intensity	y and Hydra	ulic Diameter		•	
	Turbule	ent Intensity	(%) 5				•	
	Hydrau	ılic Diameter	(in) 4				•	
			OK Can	cel Help)			

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

	(-5						
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UD
Temperatur	re (k) 293.15				-		

- Enter 293.15 [K] for Temperature.
- Click **OK** to close the **Velocity Inlet** dialog box.

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]

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	eference Frar	ne Absolute	e			•	
Gauge Pressure (pascal)								
Pressure Profile Multiplier 1								
Backflow Direction Specification Method Normal to Boundary								
Bac	kflow Pressu	re Specificati	on Total Pr	essure			•	
🗌 Radial E	quilibrium Pr	essure Distri	bution					
Average	Pressure Sp	pecification						
Target N	Mass Flow Ra	ate						
Turbu	ence							
	Specifi	cation Metho	d Intensity	and Hydrau	lic Diameter		•	
Backfl	ow Turbulen	t Intensity (%	5)				•	
Backfl	ow Hydraulio	: Diameter (ir	n) [4				•	
		[OK Can	cel Help)			

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

• Wall									×
Zone Name									
wall									
Adjacent Cell	Zone					_			
fluid									
Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential	Structure
Wall Motio	n	Motion							
 Station Moving 	ary Wall Wall	Relativ	e to Adjac	ent Cell	Zone				
Shear Con	dition								
No Slip Specifi Specifi	ed Shear								
O Marang	goni Stress	l							
Wall Rougi	nness								
Roughness	Height (in)	0							
Roughnes	ss Constan	0.5					•		
			o	Car	icel Help	Ì			
- L						ł:			

1.4.6. Solving

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

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





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

Retain the default selections.

Enable the plotting of residuals during the calculation.

In the Solution ribbon tab, click Residuals... (Reports group box).

Reports								
🖄 Residuals	¥€ Conve	rgence						
🔯 Definitions 🖕	File	<u></u> Plot						

Solution	\rightarrow	Reports	\rightarrow	Residuals
----------	---------------	---------	---------------	-----------

Residual	Monitor	Check Convergence	Absolute Criteria
12			
continuity	 Image: A start of the start of		0.001
x-velocity		v	0.001
y-velocity	V	V	0.001
z-velocity		J	0.001
energy	V	~	1e-06
k		-	0.001
omega	V	V	0.001
	x-velocity y-velocity z-velocity energy k omega	x-velocity y-velocity z-velocity energy k omega	x-velocity V V y-velocity V V z-velocity V V energy V V k V V omega V V

- Ensure that **Plot** is enabled in the **Options** group box.
- Retain the default value of 0.001 for the Absolute Criteria of continuity.
- Click **OK** to close the **Residual Monitors** dialog box.

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

			_ ~ .							-
Solution -	→ Ror	norts —	> Dofini	tions 🔿		Surface	Renort —	Macc_M	loightod	$\Delta verage$
Joiution	/ NCF	JUILS .			INC W		incport /		v cigiii cu	Average

Name	Report Type	
outlet-temp-avg	Mass-Weighted Average	*
Options	Custom Vectors	
Per Surface	Vectors of	
Average Over	Custom Vectors	
	Field Variable	
Report Files [0/0] 📃 📑	Temperature	*
	Static Temperature	-
	Surfaces (Filter Text	o E E 🗾 🖡
	default-interior	
Report Plots [0/0]	pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall	
Create		
 ✓ Report File ✓ Report Plot 		
Frequency 3		
✓ Print to Console	Highlight Surfaces	
Create Output Parameter	New Surface +	

- Enter outlet-temp-avg for the Name of the report definition.
- 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.

• 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.

- Select Temperature... and Static Temperature from the Field Variable drop-down lists.
- Select **pressure-outlet-7** from the **Surfaces** selection list.
- 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)
- 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.

Name outlet-temp-avg-rfile	Active		
Available Report Definitions [0/0]	F	Selected Report Definitions [0/1]	ج (ج
		outlet-temp-avg	10 1025
	Add>	·>	
	< <rem< td=""><td>love</td><td></td></rem<>	love	
File Name		New _ Edit	
aution-tomo-sus-file out	Browse	Control Reserves	
foner remp-avg-me.our)			
Full File Name			
Full File Name	•		

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

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.

• (optional) Edit the name and location of the resulting file as necessary using the **File Name** field or **Browse...** button.

- Click **OK** to close the **Edit Report File** dialog box.
- Create a convergence condition for **outlet-temp-avg**.

Con	vergence Conditions						×
Active	Conditions	Report Definition	Stop Criterion	Ignore Iterations Before	Use Iterations	Print	Delete
✓	con-outlet-temp-avg	outlet-temp-	1e-5	20	15	✓	
Add							
Choose	e Condition	Every Iteration					
● All ○ An	Conditions are Met y Condition is Met	3	Resid	uals			
		ок	Cancel	elp			
•	Click the Add button.						

Solution \rightarrow Reports \rightarrow Convergence...

- Enter con-outlet-temp-avg for Conditions.
- Select outlet-temp-avg from the Report Definition drop-down list.
- Enter 1e-5 for Stop Criterion.
- Enter 20 for Ignore Iterations Before.
- Enter 15 for Use Iterations.
- Enable Print.
- Set Every Iteration to 3.
- 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.

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

Solution \rightarrow Initialization

	Initia	alization	
Method		Patch	
 Hybrid Standard 	More Settings Options	──	t=0 Initialize

- Retain the default selection of Hybrid from the Method list.
- Click Initialize.

Save the case file (elbow1.cas.h5).

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

Select File									?		×
Look in:	C:\1	Tutorials\intr	oduction		•	G	0	0	B	::	
My Cor	nputer _	Name	Size	Туре	Date Modified						
Docum	ents n	L ebow.m	sn								

✓ Display Mesh After Reading

• (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.

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

Solution \rightarrow Run Calculation

	Run Calculation	
🗹 Input Summary	🃌 Run Calculation	
🛃 Check Case	Time Scale Factor 1	
♦↑ Update Dynamic Mesh	No. of Iterations 150	Calculate

- Enter 150 for No. of Iterations.
- Click Calculate.

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).



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).



Figure 1.4: Residuals

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.

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

Mass Flow Rate	Boundaries Filter Text	Denile
 Total Heat Transfer Rate Radiation Heat Transfer Rate 	default-interior pressure-outlet-7 symmetry velocity-inlet-5 velocity-inlet-6 wall	-1.915872265440582 1.614609710153648 0.3012634374729083
Save Output Parameter		Net Results (kg/s) 8.82186e-07

 $\textbf{Results} \rightarrow \textbf{Reports} \rightarrow \textbf{Fluxes}...$

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

Close the Flux Reports dialog box.

Save the data file (elbow1.dat.h5).

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

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.

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

$\textbf{Results} \rightarrow \textbf{Graphics} \rightarrow \textbf{Contours} \rightarrow \textbf{New...}$

E Contours	×
Contour Name	
contour-vel	
Options	Contours of
✓ Filled	Velocity
✓ Node Values	Velocity Magnitude
Contour Lines	Min (m/s) Max (m/s)
Clobal Range	0 1.37504
Auto Range Clip to Range Draw Profiles Draw Mesh Coloring Banded Smooth	Surfaces Filter Text Text Text Text Text Text Text Text
Colormap Options	New Surface 🐙
s	ave/Display Compute Close Help

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

Close the **Contours** dialog box.

$\mathbf{View} \rightarrow \mathbf{Display}$



Disable the Headlight and Lighting options.



Figure 1.5: Predicted Velocity Distribution after the Initial Calculation

Create and display a definition for temperature contours on the symmetry plane (<u>Figure 1.6: Predicted</u> <u>Temperature Distribution after the Initial Calculation</u>).

$\textbf{Results} \rightarrow \textbf{Graphics} \rightarrow \textbf{Contours} \rightarrow \textbf{New}...$

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

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

- Enter contour-temp for Contour Name.
- Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.
- Select **symmetry** from the **Surfaces** selection list.
- 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.

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

 $\textbf{Results} \rightarrow \textbf{Graphics} \rightarrow \textbf{Vectors} \rightarrow \textbf{New...}$



Figure 1.6: Predicted Temperature Distribution after the Initial Calculation

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

• Enter vector-vel for Vector Name.

- Select **arrow** from the **Style** drop-down box.
- Select symmetry from the Surfaces selection list.
- 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.

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



Figure 1.8: Resized Velocity Vectors

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

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



Figure 1.9: Magnified View of Resized Velocity Vectors

• 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.

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



$\textbf{Results} \rightarrow \textbf{Surface} \rightarrow \textbf{Create} \rightarrow \textbf{Iso-Surface...}$

z=0_outlet				
Surface of Cons	stant		default-interior	
Mesh		*	pressure-outlet-7	
Z-Coordinate		٣	velocity-inlet-5	
Min (in)	Max (in)		wall	
0	2		From Zones Filter Text	
Iso-Values (in) 0		fluid		
•]0			ſ	

- Enter z=0_outlet for New Surface Name.
- Select Mesh... and Z-Coordinate from the Surface of Constant drop-down lists.
- 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.
- Retain the default value of 0 inches for Iso-Values.
- Select pressure-outlet-7 from the From Surface selection list.
- 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.

Close the Iso-Surface dialog box.

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).

 $\textbf{Results} \rightarrow \textbf{Plots} \rightarrow \textbf{XY} \ \textbf{Plot} \rightarrow \textbf{New...}$

xy-outlet-temp			
Options	Plot Direct	ion Y Axis Function	
✓ Node Values	X 1	Temperature	•
Position on X Axis	Y 0	Static Temperature	
Position on Y Axis	Z [0	X Axis Function	
Write to File		Direction Vector	*
	Free Data	symmetry velocity-inlet-5 velocity-inlet-6 wall z=0_outlet	
1		New Surface	

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

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

- Click Write....
- In the Select File dialog box, enter outlet_temp1.xy for XY File.
- Click **OK** to save the temperature data and close the **Select File** dialog box.
- Close the **Solution XY Plot** dialog box.



Figure 1.10: Outlet Temperature Profile for the Initial Solution

Define a custom field function for the dynamic head formula.

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

		x	1	V^x	ABS	Select Operand Field Functions from
1			(top)		leate	Field Functions
<u> </u>	SBI	cos	tan			Velocity 👻
j	1	2	3	4	SQRT	Velocity Magnitude
	6	7	8	9	CE/C	Salect
)	PI	e		DEL	(Select)

Select **Density...** and **Density** from the **Field Functions** drop-down lists, and click the **Select** button to add **density** to the **Definition** field.

Click the **X** button to add the multiplication symbol to the **Definition** field.

Select Velocity... and Velocity Magnitude from the Field Functions drop-down lists, and click the Select button to add |V| to the Definition field.

Click **y^x** to raise the last entry in the **Definition** field to a power, and click **2** for the power.

Click the / button to add the division symbol to the **Definition** field, and then click **2**.

Enter dynamic-head for New Function Name.

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.

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

 $\textbf{Results} \rightarrow \textbf{Graphics} \rightarrow \textbf{Contours} \rightarrow \textbf{New...}$

Options	Contours of					
✓ Filled	Custom Field	d Functions				
✓ Node Values	dynamic-hea	dynamic-head				
Contour Lines	Min	Max				
	0	0				
Draw Profiles Draw Mesh Coloring Banded Smooth	symmetry velocity-inl velocity-inl wall z=0_outlet	et-5 et-6				
Colormap Options		20				

Enter contour-dynamic-head for Contour Name.

Select **Banded** in the **Coloring** group box.

Select Custom Field Functions... and dynamic-head from the Contours of drop-down lists.

Select **symmetry** from the **Surfaces** selection list.

Click Save/Display and close the Contours dialog box.



Figure 1.11: Contours of the Dynamic Head Custom Field Function

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

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

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.

Click **OK** to save the files and close the **Select File** dialog box.

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.

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

Solution \rightarrow Cell Registers New \rightarrow Field Variable...

Setup		Mesh		
General Ø Models		Scale	Check Report Qu	ality
Arrow Materials Arrow Condition	25	Display	Units	
 El Boundary Condition Dynamic Mesh 	ns	Solver		
🛃 Reference Values		Туре	Velocity Formul	lation
Keference Frames		Pressure-Based	Absolute	
Mamed Expression	S	O Density-Based	O Relative	
Solution Methods Controls				
📀 🛐 Report Definitions		lime		
🔄 🍳 Monitors		Steady		
🕑 🐻 Cell Registers	Edit	O Transient		
🚓 Initialization	Editor			
Run Calculation	New 🕨	Region		
Results	Expand All	Boundary		
💿 🧶 Surfaces	Expand All	Field Variable		
(+) Craphics	Collapse All	Limitor		
Scene		chinter		
Animations		Residuals		
Reports		Volume		
Parameters & Customiz	ation	Yplus/Ystar		
🗾 Field Variable Register				
Name curvature_0				
Туре	Curvatu	re of		
Cells More Than	 Tempe 	rature		
Derivative Option	Static	Temperature		-
Curvature	▼ Min		Max	
Scaling Option	4.93038	31e-32	0.03171117	
None	Cells ha	ving value more than	0.002	_
			0.003	

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

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

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.

Click Save and close the Field Variable Register daialog box.

Setup mesh adaption using the **Cell Registers**. For this task, you will use the **Adapt** group box in the **Domain** ribbon tab.

Adapt	
Refine / Coarsen.	
ooo More	*

 $\textbf{Domain} \rightarrow \textbf{Adapt} \rightarrow \textbf{Refine} \text{ / Coarsen...}$

Adaption Controls	×
Refinement Criterion curvature_0	•
Coarsening Criterion	•
Maximum	Refinement Level 2
Minimun	n Cell Volume (m3) 0
Dynamic Adaption	Predefined Criteria 👻
	Cell Registers 🚽
Advanced Controls	List Criteria
	Display Options
OK Adapt Display Cano	Help

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.

Click Adapt.

Click Display.

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



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) by performing the following steps:

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 ✓ Edges Centroid 	 Faces Edges Centroid 	 Faces Edges Centroid 			
	Face Color red	Face Color green	Face Color yellow Edge Color dark gray			

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

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

📕 Mesh Display		×
Options Nodes Graces Faces Overset	Edge Type All Feature Outline	Surfaces Filter Text
Shrink Factor Fe 0 2 Outline Adjacency	eature Angle 0 Interior	z=0_outlet
	Disp	olay Colors Close Help

- Ensure that only the **Edges** option is enabled in the **Options** group box.
- Select Feature from the Edge Type list.
- Select all of the items except **z=0_outlet** from the **Surfaces** selection list.
- Click **Display** and close the **Mesh Display** dialog box.
- Click **OK** to close the **Display Options Adaption** dialog box.
- Click **Display** in the **Adaption Controls** dialog box.
- Rotate the view and zoom in to get the display shown in <u>Figure 1.13: Alternative Display of Cells</u> <u>Marked for Adaption</u>.



Figure 1.13: Alternative Display of Cells Marked for Adaption

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

Click **OK** to close the **Adaption Controls** dialog box.

Display the adapted mesh (Figure 1.14: The Adapted Mesh).

options	Edge Type	
Nodes	All	Surfaces Filter Text
✓ Edges	O Feature	pressure-outlet-7
Faces	Outline	symmetry
Partitions		velocity-inlet-5
Overset		velocity-inlet-6
		wall
Shrink Factor	Feature Angle	z=o_outiet
0	20	
Outline	Interior	
)	New Surface

 $\textbf{Domain} \rightarrow \textbf{Mesh} \rightarrow \textbf{Display...}$

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

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



Figure 1.14: The Adapted Mesh

Request an additional 90 iterations.

Solution \rightarrow Run Calculation \rightarrow Calculate



The solution will converge as shown in <u>Figure 1.15: The Complete Residual</u> <u>History</u> and <u>Figure 1.16: Convergence History of Mass-Weighted Average Temperature</u>.



Figure 1.15: The Complete Residual History



Figure 1.16: Convergence History of Mass-Weighted Average Temperature

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

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

Enter elbow2.h5 for Case/Data File.

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.

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).

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





Figure 1.17: Filled Contours of Temperature Using the Adapted Mesh

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).

 $\textbf{Results} \rightarrow \textbf{Plots} \rightarrow \textbf{XY} \ \textbf{Plot} \rightarrow \textbf{xy-outlet-temp} \qquad \textbf{Edit...}$

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



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

Enable Write to File in the Options group box.

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

Click Write

In the Select File dialog box, enter outlet_temp2.xy for XY File.

Click **OK** to save the temperature data.

Close the **Solution XY Plot** dialog box.

Display the outlet temperature profiles for both solutions on a single plot (Figure 1.19: Outlet Temperature Profiles for the Two Solutions).

Open the Plot Data Sources dialog box.

 $\textbf{Results} \rightarrow \textbf{Plots} \rightarrow \textbf{Data Sources...}$

/ariables
X Axis Variable Y Axis Variables (1/2) The solution Position Static Temperature
Plot
Removel Removel Y Axis Label Static Temperature Y Axis Label Static Temperature

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

		Namo	1	Sizo		Tuno	Data Madified					
S My Cor	nputer	name		Size	757 hides	Type	tate mouned		4			
Docum	ents	outlet_tem	p1.xy		307 bytes	xy File	11/13/2018 9:27 AM					
-		outiet_tem	р2.ху	1 3	361 bytes	ху не	11/13/2018 11:20 AP	n				
-	(T)											
V Eila	outlet to										0	e .
T File	ouuer_u	траху									U	•
ilos of hunou	VV Files	(* xer * erst)								- 1	Can	col
lies of type.	AT Flies	(.xy .out)								- 13 13	Call	Cer
iltor String	(C)										rile	-
filler suring	-									-	Fat	er
										(n		
										IN	enn	ve
			marro							-		
	introducti	ion/outlet_tem	ip1.xy									
C:/Tutorials/												

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.

Click **OK** to save the files and close the **Select File** dialog box.

Select the folder path ending in **outlet_temp1.xy** from the **Curve Information** selection list (**Curves** group box).

Enter Before Adaption in the lower-right text-entry box.

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.

In a similar manner, change the legend entry for the folder path ending in **outlet_temp2.xy** to be Adapted Mesh.

Click Plot and close the Plot Data Sources dialog box.

Figure 1.19: Outlet Temperature Profiles for the Two Solutions 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.