Chapter 2: Parametric Analysis in ANSYS Workbench Using ANSYS Fluent

This tutorial is divided into the following sections:

- 2.1. Introduction
- 2.2. Prerequisites
- 2.3. Problem Description
- 2.4. Setup and Solution

2.1. Introduction

This tutorial illustrates using an ANSYS Fluent fluid flow system in ANSYS Workbench to set up and solve a three-dimensional turbulent fluid flow and heat transfer problem in an automotive heating, ventilation, and air conditioning (HVAC) duct system. ANSYS Workbench uses parameters and design points to allow you to run optimization and what-if scenarios. You can define both input and output parameters in ANSYS Fluent that can be used in your ANSYS Workbench project. You can also define parameters in other applications including ANSYS DesignModeler and ANSYS CFD-Post. Once you have defined parameters for your system, a **Parameters** cell is added to the system and the **Parameter Set** bus bar is added to your project. This tutorial is designed to introduce you to the parametric analysis utility available in ANSYS Workbench.

The tutorial starts with a Fluid Flow (Fluent) analysis system with pre-defined geometry and mesh components. Within this tutorial, you will redefine the geometry parameters created in ANSYS Design-Modeler by adding constraints to the input parameters. You will use ANSYS Fluent to set up and solve the CFD problem. While defining the problem set-up, you will also learn to define input parameters in ANSYS Fluent. The tutorial will also provide information on how to create output parameters in ANSYS CFD-Post.

This tutorial demonstrates how to do the following:

- Add constraints to the ANSYS DesignModeler input parameters.
- Create an ANSYS Fluent fluid flow analysis system in ANSYS Workbench.
- Set up the CFD simulation in ANSYS Fluent, which includes:
 - Setting material properties and boundary conditions for a turbulent forced convection problem.
 - Defining input parameters in Fluent
- Define output parameters in CFD-Post
- Create additional design points in ANSYS Workbench.
- Run multiple CFD simulations by updating the design points.

• Analyze the results of each design point project in ANSYS CFD-Post and ANSYS Workbench.

Important

The mesh and solution settings for this tutorial are designed to demonstrate a basic parameterization simulation within a reasonable solution time-frame. Ordinarily, you would use additional mesh and solution settings to obtain a more accurate solution.

2.2. Prerequisites

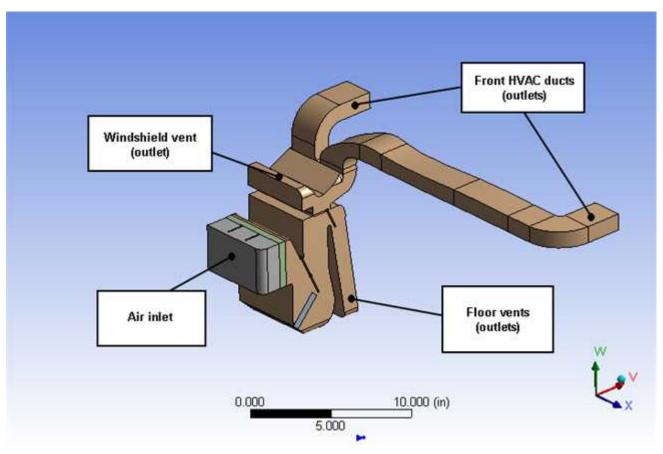
This tutorial assumes that you are already familiar with the ANSYS Workbench interface and its project workflow (for example, ANSYS DesignModeler, ANSYS Meshing, ANSYS Fluent, and ANSYS CFD-Post). This tutorial also assumes that you have completed Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1), and that you are familiar with the ANSYS Fluent graphical user interface. Some steps in the setup and solution procedure will not be shown explicitly.

2.3. Problem Description

In the past, evaluation of vehicle air conditioning systems was performed using prototypes and testing their performance in test labs. However, the design process of modern vehicle air conditioning (AC) systems improved with the introduction of Computer Aided Design (CAD), Computer Aided Engineering (CAE) and Computer Aided Manufacturing (CAM). The AC system specification will include minimum performance requirements, temperatures, control zones, flow rates, and so on. Performance testing using CFD may include fluid velocity (air flow), pressure values, and temperature distribution. Using CFD enables the analysis of fluid through very complex geometry and boundary conditions.

As part of the analysis, a designer can change the geometry of the system or the boundary conditions such as the inlet velocity, flow rate, and so on, and view the effect on fluid flow patterns. This tutorial illustrates the AC design process on a representative automotive HVAC system consisting of both an evaporator for cooling and a heat exchanger for heating requirements. This HVAC system is symmetric, so the geometry has been simplified using a plane of symmetry to reduce computation time.





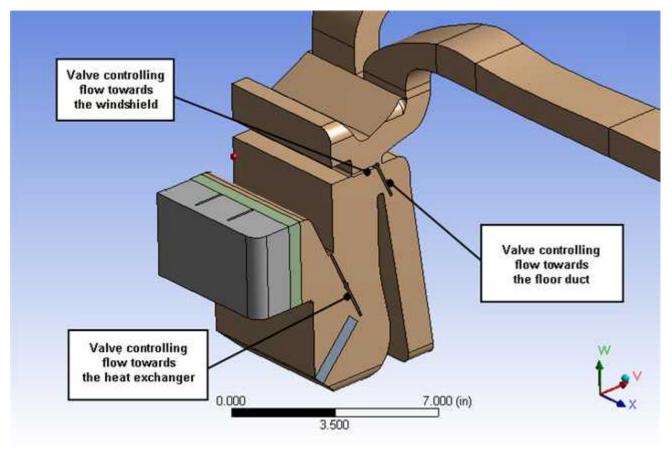


Figure 2.2: HVAC System Valve Location Details

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

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

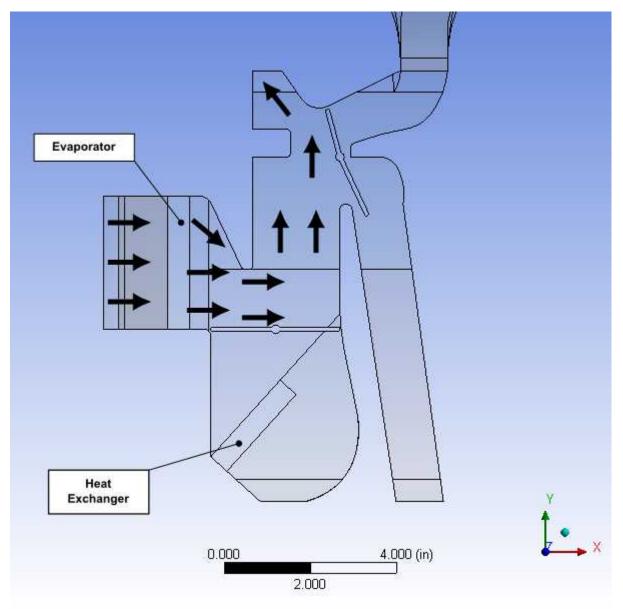
Air enters the HVAC system at 310 K with a velocity of 0.5 m/sec through the air inlet and passes to the evaporator and then, depending on the position of the valve controlling flow to the heat exchanger, flows over or bypasses the heat exchanger. Depending on the cooling and heating requirements, either the evaporator or the heat exchanger would be operational, but not both at the same time. The position of the other two valves controls the flow towards the front panel, the windshield, or towards the floor ducts.

The motion of the valves is constrained. The valve controlling flow over the heat exchanger varies between 25° and 90°. The valve controlling the floor flow varies between 20° and 60°. The valve controlling flow towards front panel or windshield varies between 15° and 175°.

The evaporator load is about 200 W in the cooling cycle. The heat exchanger load is about 150 W.

This tutorial illustrates the easiest way to analyze the effects of the above parameters on the flow pattern/distribution and the outlet temperature of air (entering the passenger cabin). Using the parametric analysis capability in ANSYS Workbench, a designer can check the performance of the system at various design points.

Figure 2.3: Flow Pattern for the Cooling Cycle



2.4. 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. xvi) for detailed information.

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

2.4.1. Preparation
2.4.2. Adding Constraints to ANSYS DesignModeler Parameters in ANSYS Workbench
2.4.3. Setting Up the CFD Simulation in ANSYS Fluent
2.4.4. Defining Input Parameters in ANSYS Fluent
2.4.5. Solving

2.4.6. Postprocessing and Setting the Output Parameters in ANSYS CFD-Post

2.4.7. Creating Additional Design Points in ANSYS Workbench 2.4.8. Postprocessing the New Design Points in CFD-Post 2.4.9. Summary

2.4.1. Preparation

To prepare for running this tutorial:

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

Note

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

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
 - a. Click ANSYS Fluent under Product.
 - b. Click **18.0** under **Version**.
- 5. Select this tutorial from the list.
- 6. Click the **workbench-parameter-tutorial_R180.zip** link to download the input files.
- 7. Unzip the workbench-parameter-tutorial_R180.zip file to your working folder.

The extracted workbench-parameter-tutorial folder contains a single archive file fluentworkbench-param.wbpz that includes all supporting input files of the starting ANSYS Workbench project and a folder called final_project_files that includes the archived final version of the project. The final result files incorporate ANSYS Fluent and ANSYS CFD-Post settings and all already defined design points (all that is required is to update the design points in the project to generate corresponding solutions).

Note

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

2.4.2. Adding Constraints to ANSYS DesignModeler Parameters in ANSYS Workbench

In this step, you will start ANSYS Workbench, open the project file, review existing parameters, create new parameters, and add constraints to existing ANSYS DesignModeler parameters.

1. From the Windows Start menu, select Start > All Programs > ANSYS 18.0 > Workbench 18.0 to start ANSYS Workbench.

This displays the ANSYS Workbench application window, which has the **Toolbox** on the left and the **Project Schematic** to its right. Various supported applications are listed in the **Toolbox**, and the components of the analysis system are displayed in the **Project Schematic**.

Note

When you first start ANSYS Workbench, the **Getting Started** message window is displayed, offering assistance through the online help for using the application. You can keep the window open, or close it by clicking **OK**. If you need to access the online help at any time, use the **Help** menu, or press the **F1** key.

2. Restore the archive of the starting ANSYS Workbench project to your working directory.

File \rightarrow Restore Archive...

The Select Archive to Restore dialog box appears.

a. Browse to your working directory, select the project archive file fluent-workbench-param.wbpz, and click **Open**.

The Save As dialog box appears.

b. Browse, if necessary, to your working folder and click **Save** to restore the project file, fluent-workbench-param.wbpj, and a corresponding project folder, fluent-workbench-param_files, for this tutorial.

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

A fluent-workbench-param - Workbench	
And the second frames and the second	Jobs Help
	ect 🍠 Update Project 朔 Update All Design Points 🌉 ACT Start F
Toolbox 🔹 🕂 🗙	Project Schematic
Analysis Systems	
Fluid Flow (Fluent)	▼ A
→ Throughflow	1 G Fluid Flow (FLUENT)
Throughflow (BladeGen)	2 🔀 Geometry
Component Systems	3 🍘 Mesh 🗸
Design Exploration	4 🍓 Setup 😨
☑ External Connection Systems	5 😭 Solution 👕
	6 🗑 Results
	→ 7 🛱 Parameters
	Fluid Flow (FLUENT)
	Parameter Set
View All / Customize	
Ready	🚮 Job Monitor 🔟 Show Progress

Figure 2.4: The Project Loaded into ANSYS Workbench

The project (fluent-workbench-param.wbpj) already has a Fluent-based fluid flow analysis system that includes the geometry and mesh, as well as some predefined parameters. You will first examine and edit parameters within Workbench, then later proceed to define the fluid flow model in ANSYS Fluent.

3. Open the **Files** view in ANSYS Workbench so you can view the files associated with the current project and are written during the session.

 $\textbf{View} \rightarrow \textbf{Files}$

Analysis Systems Fluid Flow (Fluent) I C Engine (Fluent) Throughflow Throughflow (BladeGen) Component Systems Design Exploration External Connection Systems Fluid Flow (FLUENT) S S Results Fluid Flow (FLUENT) Fluid Flow (FLUEN		Conception in the	020103		7.04599622014-00	15315	ACT Start P	157		_
Fluid Flow (Fluent) 1 CEngine (Fluent) Throughflow Throughflow (BladeGen) Component Systems Design Exploration External Connection Systems S @ Setup S @ Setup S @ Setup S @ Results Fluid Flow (FLUENT) S @ Geometry et al. S	oolbex 👻 🛡 🗙	Project	t Scher	natic						
I C Engine (Fluent) Throughflow Throughflow (BladeGen) Component Systems Design Exploration External Connection Systems	E Analysis Systems	1								
In Explant (Vertex) Throughlow (BladeGen) Component Systems Design Exploration External Connection Systems State Solution State Solution File A B C Design Explore and the	Fluid Flow (Fluent)		1027							
Throughflow (BladeGen) Component Systems Design Exploration External Connection Systems S @ Solution Files Files Image: Solution Image: Solution Image: Solution Files Image: Solution					-					
Component Systems Design Exploration External Connection Systems Setup Parameters Fluid Flow (FLUENT) Fles A B C Description Setup Name Ce Size Type Setup Setup Setup Name Setup Setup Setup Name Setup Setup Setup Setup					anj -					
Design Exploration External Connection Systems 5 Solution 6 Results 7 Parameters Fluid Flow (FLUENT) Image: Parameter Set Fluid Flow (FLUENT) Fluid Flow (FLUENT) Image: Parameter Set Imag		Ū.		a supply products	· .					
External Connection Systems			3 1	Mesh	а,					
Solution Image: Solution 6 Results 7 Parameters Fluid Flow (FLUENT) Parameter Set			4 8	🔍 Setup						
7 7 <th7< th=""> <th7< th=""> <th7< th=""></th7<></th7<></th7<>	1 External Connection Systems		5 6	Solution	°.					
Fluid Flow (FLUENT) Fluid Flow (FLUENT) Fles Fles A B C D 1 Name Ce Size Type 2 A fluent-workbench-param.wbpj 134 K8 Workbench Project! 3 @ Geom.agdb A2 2 2M8 Geometry File			6 6	Results	2.					
Fluid Flow (FLUENT) Fluid Flow (FLUENT) Fles Fles A B C D 1 Name Ce Size Type 2 A fluent-workbench-param.wbpj 134 K8 Workbench Project! 3 @ Geom.agdb A2 2 2M8 Geometry File										
I Name Ce Size Type 2 A fluent-workbench-param.wbpi 134 KB Workbench Project 3 Geom.agdb A2 2 MB				Fluid Flow (FLU	NT)					
2 A fluent-workbench-param.wbpj 134 KB Workbench Project 3 Geom.agdb A2 2 MB Geometry File		द्य		Fluid Flow (FLUE	NT)					
3 Geom.agdb A2 2MB Geometry File		Files		Fluid Flow (FLUE eter Set	NT)		10000	1.121.12	- Phili	
		Files		Fluid Flow (FLUE eter Set A Name			10000	Size 💌	Туре	
and		Files 1 2	Param	Fluid Flow (FLUE eter Set A Name fluent-workbench-			Ce *	Size • 134 KB	Type Workbench Pre	oject F
5 FFF.mshdb A3 184.KB mshdb		(57) Files	Param	Fluid Flow (FLU) eter Set A Name fluent-workbench- Geom.agdb		-	Ce * A2	Size • 134 KB 2 MB	Type Warkbendh Pri Geometry File	oject F
		(5) Files 1 2 3 4	Param	Fluid Flow (FLU) eter Set A Name fluent-workbench- Geom.agdb FFF.agdb		-	Ce • A2 A2	Size	Type Warkbench Pro Geometry File Geometry File	oject F
6 desgnPoint.wbdp 37.KB Workbench Design P		Files 1 2 3 4 5	Param	Fluid Flow (FLUE eter Set A Name fluent-workbench- Geom.agdb FFF.agdb FFF.agdb			Ce * A2	Size • 134 K8 2 M8 2 M8 184 K8	Type Warkbench Pri Geometry File Geometry File .mshdb	
		द्य		Fluid Flow (FLU	NT)		B	C	D	
		Files 1 2 3 4 5	Param	Fluid Flow (FLUE eter Set A Name fluent-workbench- Geom.agdb FFF.agdb FFF.agdb			Ce • A2 A2	Size • 134 K8 2 M8 2 M8 184 K8	Type Warkbench Pri Geometry File Geometry File .mshdb	oject i
6 designPoint.wbdp 37 KB Workbench Design F 7 DesignPointLog.csv 2 KB .csv		Files 1 2 3 4 5 6	Param	Fluid Flow (FLUE eter Set A Name fluent-workbench- Geom.agdb FFF.agdb FFF.agdb FFF.agdb FFF.mshdb	aram.wbpj		Ce • A2 A2	Size • 134 KB 2 MB 2 MB 184 KB 37 KB	Type Workbench Pri Geometry File Jesometry File Jesometry File Workbench De	oject P

Figure 2.5: The Project Loaded into ANSYS Workbench Displaying Properties and Files View

Note the types of files that have been created for this project. Also note the states of the cells for the Fluid Flow (Fluent) analysis system. Since the geometry has already been defined, the status of the **Geometry** cell is Up-to-Date (\checkmark). Since the mesh is not complete, the **Mesh** cell's state is Refresh Required (\gtrless), and since the ANSYS Fluent setup is incomplete and the simulation has yet to be performed, with

no corresponding results, the state for the **Setup**, **Solution**, and **Results** cells is Unfulfilled (\mathbb{P}). For more information about cell states, see the Workbench User's Guide.

- 4. Review the input parameters that have already been defined in ANSYS DesignModeler.
 - a. Double-click the **Parameter Set** bus bar in the ANSYS Workbench **Project Schematic** to open the **Parameters Set** tab.

Note

To return to viewing the **Project Schematic**, click the **Project** tab.

- b. In the **Outline of All Parameters** view (Figure 2.6: Parameters Defined in ANSYS DesignModeler (p. 82)), review the following existing parameters:
 - The parameter hcpos represents the valve position that controls the flow over the heat exchanger. When the valve is at an angle of 25°, it allows the flow to pass over the heat exchanger. When the angle is 90°, it completely blocks the flow towards the heat exchanger. Any value in between allows some flow to pass over the heat exchanger giving a mixed flow condition.
 - The parameter ftpos represents the valve position that controls flow towards the floor duct. When the valve is at an angle of 20°, it blocks the flow towards the floor duct and when the valve angle is 60°, it unblocks the flow.
 - The parameter wsfpos represents the valve position that controls flow towards the windshield and the front panel. When the valve is at an angle of 15°, it allows the entire flow to go towards the windshield. When the angle is 90°, it completely blocks the flow towards windshield as well as the front panel. When the angle is 175°, it allows the flow to go towards the windshield and the front panel.

	А	В	с	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	🗏 🖾 Fluid Flow (FLUENT) (A1)			
4	ί <mark>ρ</mark> Ρ1	hcpos	90	
5	ί <mark>ρ</mark> Ρ2	ftpos	25	
6	🗘 РЗ	wsfpos	175	
*	🗘 New input parameter	New name	New expression	
8	Output Parameters			
*	🔁 New output parameter		New expression	
10	Charts			

Figure 2.6: Parameters Defined in ANSYS DesignModeler

- 5. In the **Outline of All Parameters** view, create three new named input parameters.
 - a. In the row that contains **New input parameter**, click the parameter table cell with **New name** (under the **Parameter Name** column) and enter **input_hcpos**. Note the ID of the parameter that appears in column **A** of the table. For the new input parameter, the parameter **ID** is **P4**. In the **Value** column, enter **15**.
 - b. In a similar manner, create two more parameters named input_ftpos and input_wsfpos. In the Value column, enter 25, and 90 for each new parameter (P5 and P6), respectively.

	А	В	с	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	🖃 区 Fluid Flow (FLUENT) (A1)			
4	ι <mark>ρ</mark> Ρ1	hcpos	90	
5	ι <mark>ρ</mark> Ρ2	ftpos	25	
6	🗘 РЗ	wsfpos	175	
7	🗘 Р4	input_hcpos	15	
8	🗘 Р5	input_ftpos	25	
9	🗘 Рб	input_wsfpos	90	
*	🏟 New input parameter	New name	New expression	
11	Output Parameters			
*	🔁 New output parameter		New expression	
13	Charts			

Figure 2.7: New Parameters Defined in ANSYS Workbench

6. Select the row (or any cell in the row) that corresponds to the hcpos parameter. In the Properties of Outline view, change the value of the hcpos parameter in the Expression field from 90 to the expression min(max(25,P4),90). This puts a constraint on the value of hcpos, so that the value always remains between 25° and 90°. The redefined parameter hcpos is automatically passed to ANSYS DesignModeler. Alternatively the same constraint can also be set using the expression max(25, min(P4,90)).

After defining this expression, the parameter becomes a derived parameter that is dependent on the value of the parameter input_hcpos with ID **P4**. The derived parameters are unavailable for editing in the **Outline of All Parameters** view and could be redefined only in the **Properties of Outline** view.

Important

When entering expressions, you must use the list and decimal delimiters associated with your selected language in the Workbench regional and language settings, which correspond to the regional settings on your machine. The instructions in this tutorial assume that your systems uses "." as a decimal separator and "," as a list separator.

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

Outline	of All Parameters 🔹 📮 🗙						
	А		В	с			
1	ID		Parameter Name	Value			
2	Input Parameters						
3	🖃 🙆 Fluid Flow (FLU	IENT) (A1)					
4	ι <mark>ρ</mark> Ρ1		hcpos	25			
5	🛱 P2		ftpos	25			
6	🗘 P3		wsfpos	175			
7	ί <mark>ρ</mark> Ρ4		input_hcpos	15			
8	ί <mark>ρ</mark> Ρ5		input_ftpos	25			
9	🗘 P6		input_wsfpos	90			
*	🗘 New input para	ameter	New name	New expression			
11	Output Parameters						
*	😡 New output pa	rameter		New expression			
13	Charts						
•				4			
Properti	operties of Outline A4: P1 모 구 🗙						
	A	В					
1	Property		Value				
2	General						
3	Expression	min(max(25,P4),90)					
4	Description						
5	Error Message						
6	Expression Type	Derived					
7	Usage	Input					
8	Quantity Name	Dimension	ess				

 Select the row or any cell in the row that corresponds to the ftpos parameter and create a similar expression for ftpos:min(max(20, P5), 60).

Outline of All Parameters **ч** д х С A В ID Parameter Name Value 1 Input Parameters 2 Fluid Flow (FLUENT) (A1) 3 ίρ P1 4 hcpos 25 🕻 P2 5 ftpos 25 🕻 P3 6 wsfpos 175 🕻 P4 7 input_hcpos 15 8 🕻 P5 input_ftpos 25 ۲þ. 9 P6 input_wsfpos 90 * New input parameter New name New expression Output Parameters 11 -* New output parameter New expression 13 Charts Ш < □ Þ Properties of Outline B5: P2 **⊸ д х** Α в 1 Property Value General 2 3 Expression min(max(20,P5),60) 4 Description 5 Error Message 6 Expression Type Derived 7 Usage Input 8 Quantity Name Dimensionless

Figure 2.9: Constrained Parameterftpos

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

Outline of	line of All Parameters 🔹 🔻 🕇					
	А		В	с		
1	ID		Parameter Name	Value		
2	Input Parameters					
3	🖃 🙆 Fluid Flow (FLU	ENT) (A1)				
4	ι <mark>ρ</mark> Ρ1		hcpos	25		
5	ι <mark>φ</mark> Ρ2		ftpos	25		
6	ί <mark>ρ</mark> Ρ3		wsfpos	90		
7	ί <mark>ρ</mark> Ρ4		input_hcpos	15		
8	ί <mark>ρ</mark> Ρ5		input_ftpos	25		
9	ί <mark>ρ</mark> Ρ6		input_wsfpos	90		
*	🗘 New input para	ameter	New name	New expression		
11	Output Parameters					
*	Rew output pa	rameter		New expression		
13	Charts					
•				•		
Propertie	es of Outline C6: P3			~ д Х		
	A		В			
1	Property		Value			
2	General					
3	Expression	min(max(1	5,P6),175)			
4	Description					
5	Error Message					
6	Expression Type	Derived				
7	Usage	Input				
8	Quantity Name	Dimension	ess			

9. Click the **X** on the right side of the **Parameters Set** tab to close it and return to the **Project Schematic**.

Note the new status of the cells in the Fluid Flow (Fluent) analysis system. Since we have changed the values of hcpos, ftpos, and wsfpos to their new expressions, the **Geometry** and **Mesh** cells now indicates Refresh Required (\gtrless).

10. Update the **Geometry** and **Mesh** cells.

- a. Right-click the **Geometry** cell and select the **Update** option from the context menu.
- b. Likewise, right-click the **Mesh** cell and select the **Refresh** option from the context menu. Once the cell is refreshed, then right-click the **Mesh** cell again and select the **Update** option from the context menu.
- 11. Save the project in ANSYS Workbench.

In the main menu, select File \rightarrow Save

2.4.3. Setting Up the CFD Simulation in ANSYS Fluent

Now that you have edited the parameters for the project, you will set up a CFD analysis using ANSYS Fluent. In this step, you will start ANSYS Fluent, and begin setting up the CFD simulation.

2.4.3.1. Starting ANSYS Fluent

In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the ANSYS Fluent fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

When ANSYS Fluent is first started, Fluent Launcher is displayed, allowing you to view and/or set certain ANSYS Fluent start-up options.

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

Figure 2.11: ANSYS Fluent Launcher

E Fluent Launcher (Setting Edit Only)					
ANSYS	Fluent Launcher				
Dimension © 2D @ 3D	Options Double Precision Meshing Mode				
Display Options ☑ Display Mesh After Reading ☑ Workbench Color Scheme □ Do not show this panel again	Processing Options Serial Parallel				
ACT Option					
Show More Options					
OK Cancel Help -					

1. Ensure that the proper options are enabled.

Important

Note that the **Dimension** setting is already filled in and cannot be changed, since ANSYS Fluent automatically sets it based on the mesh or geometry for the current system.

a. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

Note

An option is enabled when there is a check mark in the check box, and disabled when the check box is empty. To change an option from disabled to enabled (or vice versa), click the check box or the text.

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

Note

Parallel processing offers a substantial reduction in computational time. Refer to Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 121) in this manual and the Fluent User's Guide for further information about using the parallel processing capabilities of ANSYS Fluent.

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

Note

Fluent will retain your preferences for future sessions.

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

0 0 0 0 0 1 A 1	7 12							
Péo 🌒 Setting Up Domain	de Setting Up Physics	s User Defined	Solving 🕑 Posts	pricessing	Viewing Parallel	Design	0	08. MIG
Drift Display	in the second	Combine Delete Separate Delete Adjacency Activat	Append te Replace Nesh	Interfaces	Hesh Models Dynamic Nesh Maing Planes Turbo Topology	Adapt Mark/Actast Cells Manage Regators More	Surface	
Tele /	Task Page		S	Me	eh :			
Personal Second Se	Healt Scale Declar Solve Tope Pressure-Based Declary-Based Declary-Based Tome Staady Tome Staady Tome Staady Tome Staady Tome Staady Tome Healt	veck Report Quality Welocity Formulation Absolute Rolotive	Setting wall-hes Setting wall-com Setting symmetry Setting symmetry Setting symmetry Setting interior	t-exchanger tral-unit in -destral-uni -avapotator -heat-faichur -fluid-canti -fluid-canti	cal-unit-fluid-evap	ne.		به
			Done.					4

Figure 2.12: The ANSYS Fluent Application

2.4.3.2. Setting Up Physics

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

S	Setting Up Physics → Solver						
File	🍓 Setting Up Doma	ain 🍓	Setting Up I	hysics	User Defined 🧃		
		Solve	er				
Time Stea	Type ady		y Formulation Absolute	Operati	ng Conditions		
Trar	nsient 🔘 Density-Bas	sed 🔘 R	elative < Reference Values		ence Values		

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

	Radiation	Nultiphase	🙆 Solidify/Melt			
Energy	[≫] _≠ Heat Exchanger	🔿 Species	句) Acoustics			
	🕓 Viscous	📑 Discrete Phase	🗄 More 🖕			
Models						

a. Enable heat transfer by activating the energy equation.

In the Setting Up Physics ribbon tab, select Energy (Models group).



b. Enable the k- ε turbulence model.

Setting Up Physics → Mode	ls → Viscous
U iscous Model	
Model	Model Constants
Inviscid	Cmu 🔺
🔘 Laminar	0.09
Spalart-Allmaras (1 eqn)	C1-Epsilon
k-epsilon (2 eqn)	1.44
 k-omega (2 eqn) Transition k-kl-omega (3 eqn) 	C2-Epsilon
 Transition SST (4 eqn) 	1.92
Reynolds Stress (7 eqn)	TKE Prandtl Number
Scale-Adaptive Simulation (SAS)	1
Detached Eddy Simulation (DES)	TDR Prandtl Number
Carge Eddy Simulation (LES)	1.3
k-epsilon Model	Energy Prandtl Number
Standard	0.85
© RNG	Wall Prandtl Number 🗸 🗸
Realizable	L]
Near-Wall Treatment	User-Defined Functions
Standard Wall Functions	Turbulent Viscosity
Scalable Wall Functions	none 🔻
 Non-Equilibrium Wall Functions Enhanced Wall Treatment 	Prandtl Numbers
Menter-Lechner	TKE Prandtl Number
 User-Defined Wall Functions 	none
Enhanced Wall Treatment Options	TDR Prandtl Number
Pressure Gradient Effects	none
Thermal Effects	Energy Prandtl Number
	none
Options	Wall Prandtl Number
 Viscous Heating Curvature Correction 	none
Production Kato-Launder	
Production Limiter	
OK (Cancel Help

- i. Select **k-epsilon (2 eqn)** from the **Model** group box.
- ii. Select Enhanced Wall Treatment from the Near-Wall Treatment group box.

The default Standard Wall Functions are generally applicable when the cell layer adjacent to the wall has a y+ larger than 30. In contrast, the Enhanced Wall Treatment option provides consistent solutions for all y+ values. Enhanced Wall Treatment is recommended when using the k-epsilon model for general single-phase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model refer to the Fluent User's Guide.

iii. Click **OK** to retain the other default settings, enable the model, 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.

3. Define a heat source cell zone condition for the evaporator volume.

\blacksquare Setting Up Physics \rightarrow Zones \rightarrow Cell Zones

Note

All cell zones defined in your simulation are listed in the **Cell Zone Conditions** task page and under the **Setup/Cell Zone Conditions** tree branch.

a. In the **Cell Zone Conditions** task page, under the **Zone** list, select **fluid-evaporator** and click **Edit...** to open the **Fluid** dialog box.

E Fluid								
Zone Name								
fluid-evaporator		1990 B 1 1 22						
Material Name air		• Edit						
Frame Motion	3D Fan Zone 🔋	Source Terms						
🔝 Metan 🔟	Laminar Zone	Fixed Values						
🗇 Porous Zone								
Reference Frame	Nesh Mation	Porous Zónie	30 fild Zóhli	Embedded LES	:Reaction!	Source Terms	Fixed Values	Multiplisase
-	Mass 0 so	urces	Edit					
	Momentum 0 so		Edit					
			And the second s					
Y	Momentum 0 so	urces	Edit					
2	Momentum 0 so	urces	Edit					
Turbulent Ki	netic Energy 0 so	urces	Edit					
Turbulent Dis	spation Rate 0 so	urces	Edition					
	Energy 1 so	urce	Edit					
			(100)	(a) (1)				
			OK	Cancel Help				

- b. In the Fluid dialog box, enable Source Terms.
- c. In the **Source Terms** tab, click the **Edit...** button next to **Energy**.

Energy sources	
	Number of Energy sources 1
1. (w/m3) -787401.6	⊂onstant ▼
ОК Са	ncel Help

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

2.4.4. Defining Input Parameters in ANSYS Fluent

You have now started setting up the CFD analysis using ANSYS Fluent. In this step, you will define boundary conditions and input parameters for the velocity inlet.

- 1. Define an input parameter called in_velocity for the velocity at the inlet boundary.
 - a. In the Setting Up Physics tab, click Boundaries (Zones group).

Setting Up Physics \rightarrow Zones \rightarrow Boundaries

This opens the Boundary Conditions task page.

Note

All boundaries defined in the case are also displayed under the **Setup/Boundary Conditions** tree branch.

- b. In the Boundary Conditions task page, click the Toggle Tree View button (in the upper right corner), and under the Group By category, select Zone Type. This displays boundary zones grouped by zone type.
- c. Under the Inlet zone type, double-click inlet-air.

💶 Velocity Inle	t						×
Zone Name							
inlet-air							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Velocit	y Specification I	Method Ma	gnitude, No	rmal to B	oundary		•
	Reference	Frame Ab	solute				•
	Velocity Magr	nitude (m/s	0.5		in_ve	locity	•
Supersonic/In	itial Gauge Press	sure (pascal) 0		const	tant	-
	- Turbulence						
	Specification M	lethod Inte	ensity and H	ydraulic D)iameter		•
			Turbulent I	Intensity	(%) 5		P
			Hydraulic	Diameter	(m) 0.061		P
		OK	Cancel	Help			

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

This displays the Input Parameter Properties dialog box.

💶 Input Parameter Properties 🛛 🗾	
Name	
in_velocity	-
Current Value (m/s)	
0.5	
Used In:	
	1
OK Cancel Help	

- e. Enter in_velocity for the Name, and enter 0.5 m/s for the Current Value.
- f. Click OK to close the Input Parameter Properties dialog box.
- g. Under the **Turbulence** group box, from the **Specification Method** drop-down list, select **Intensity** and Hydraulic Diameter.
- h. Retain the value of 5 % for **Turbulent Intensity**.

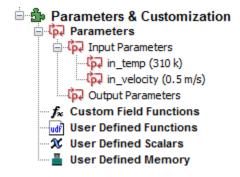
- i. Enter 0.061 for **Hydraulic Diameter (m)**.
- 2. Define an input parameter called in_temp for the temperature at the inlet boundary.

Velocity Inlet							×
Zone Name							
inlet-air							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Temperature (k)	310		in_tem	р	-		
OK Cancel Help							

- a. In the **Thermal** tab of the **Velocity Inlet** dialog box, select **New Input Parameter...** from the **Temperature** drop-down list.
- b. Enter in_temp for the Name and enter 310 K for the Current Value in the Input Parameter Properties dialog box.
- c. Click OK to close the Input Parameter Properties dialog box.
- d. Click **OK** to close the **Velocity Inlet** dialog box.
- 3. Review all of the input parameters that you have defined in ANSYS Fluent under the **Parameters & Cus-tomization/Parameters** tree branch.

► Parameters & Customization → Parameters → Input Parameters

Figure 2.13: The Input Parameters Sub-Branch in ANSYS Fluent



These parameters are passed to ANSYS Fluent component system in ANSYS Workbench and are available for editing in ANSYS Workbench (see Figure 2.14: The Parameters View in ANSYS Workbench (p. 95)).

Outline	Outline of All Parameters 🗸 🗸 🦷										
	А	В	с	D							
1	ID	Parameter Name	Value	Unit							
2	Input Parameters										
3	🖃 🔯 Fluid Flow (FLUENT) (A1)										
4	ြီး P1	hcpos	25								
5	ί <mark>ρ</mark> Ρ2	ftpos	25								
6	ြို့ P3	wsfpos	90								
7	ί <mark>ρ</mark> Ρ7	in_velocity	0.5	m s^-1 💌							
8	<mark>ф</mark> Р9	in_temp	310	к 💌							
9	С <mark>р</mark> Р4	input_hcpos	15								
10	<mark>Ґр</mark> Р5	input_ftpos	25								
11	🛱 Рб	input_wsfpos	90								
*	🗘 New input parameter	New name	New expression								
13	Output Parameters										
*	New output parameter		New expression								
15	Charts										

Figure 2.14: The Parameters View in ANSYS Workber	ch
---	----

- 4. Set the turbulence parameters for backflow at the front outlets and foot outlets.
 - a. In the **Boundary Conditions** task page, type **outlet** in the **Zone** filter text entry field. Note that as you type, the names of the boundary zones beginning with the characters you entered appear in the boundary condition zone list.

Note

- The search string can include wildcards. For example, entering ***let*** will display all zone names containing let, such as inlet and outlet.
- To display all zones again, click the red **X** icon in the **Zone** filter.
- b. Double-click **outlet-front-mid**.

	Pressure Outle	t						— ×
Z	one Name							
0	utlet-front-mid							
Γ	Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
	Bac	ckflow Refe	rence Frame	Absolute				•
		Gauge P	Pressure (paso	al) 0		CC	onstant	•
E	Backflow Direction	on Specifica	tion Method	Normal to B	oundary			•
	Backflov	w Pressure	Specification (Total Pressu	ire			•
	🗌 Radial Equilibr	rium Pressur	re Distribution					
	Average Pres	sure Specif	ication					
	Target Mass F	Flow Rate						
		Turbulen	ce					
		Specificat	ion Method []	ntensity and	d Hydraul	ic Diameter		•
			Backflo	w Turbulen	t Intensi	ty (%) 5		P
			Backf	low Hydraul	ic Diamet	er (m) 0.044		P
L								
	OK Cancel Help							

- c. In the **Pressure Outlet** dialog box, under the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.
- d. Retain the value of 5 for **Backflow Turbulent Intensity (%)**.
- e. Enter 0.044 for Backflow Hydraulic Diameter (m).

These values will only be used if reversed flow occurs at the outlets. It is a good idea to set reasonable values to prevent adverse convergence behavior if backflow occurs during the calculation.

- f. Click OK to close the Pressure Outlet dialog box.
- g. Copy the boundary conditions from **outlet-front-mid** to the other front outlet.

F Setup \rightarrow Boundary Conditions \rightarrow outlet-front-mid $\stackrel{\textcircled{U}}{\rightarrow}$ Copy...

rom Boundary Zone Filter Text	To Boundary Zones Filter Text
interior-fluid-central-unit-fluid-heat-excha 🔺	outlet-foot-left
interior-fluid-evaporator	outlet-front-side-left
interior-fluid-heat-exchanger	outlet-windshield
 Outlet 	
outlet-foot-left	
outlet-front-mid	
outlet-front-side-left	
outlet-windshield	
	11:

- i. Confirm that outlet-front-mid is selected in the From Boundary Zone selection list.
- ii. Select outlet-front-side-left in the To Boundary Zones selection list.
- iii. Click **Copy** to copy the boundary conditions.

Fluent will display a dialog box asking you to confirm that you want to copy the boundary conditions.

- iv. Click **OK** to confirm.
- v. Close the Copy Conditions dialog box.
- h. In a similar manner, set the backflow turbulence conditions for **outlet-foot-left** using the values in the following table:

Parameter	Value
Specification Method	Intensity and Hydraulic Diameter
Backflow Turbulent Intensity (%)	5
Backflow Hydraulic Diameter (m)	0.052

2.4.5. Solving

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

Note

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

1. Set the Solution Methods.





This will open the **Solution Methods** task page.

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled 👻	
Spatial Discretization	
Gradient	1
Least Squares Cell Based 🔹	
Pressure	
PRESTO!	
Momentum	Ξ
First Order Upwind 🔹	
Turbulent Kinetic Energy	
First Order Upwind 🔹	
Turbulent Dissipation Rate	T
First Order Upwind 🔹	
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Default	
Help	

a. From the **Scheme** drop-down list, select **Coupled**.

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

b. In the **Spatial Discretization** group box, configure the following settings:

Setting	Value
Pressure	PRESTO!
Momentum	First Order Upwind
Energy	First Order Upwind

This tutorial is primarily intended to demonstrate the use of parameterization and design points when running Fluent from Workbench. Therefore, you will run a simplified analysis using first order discretization, which will yield faster convergence. These settings were chosen to speed up solution time for this tutorial. Usually, for cases like this, we would recommend higher order discretization settings to be set for all flow equations to ensure improved results accuracy.

2. Initialize the flow field using the **Initialization** group of the **Solving** ribbon tab.

	Initializ	ation	
Method		Patch	
O Hybrid	More Settings	Reset Statistics	
Standard	Options	195955 91010050	t = 0
The states and states	op all marries	Reset DPM	Initialize

- a. Retain the default selection of Hybrid Initialization.
- b. Click the **Initialize** button.

Solving \rightarrow Initialization

3. Run the simulation in ANSYS Fluent from the Run Calculation group of the Solving tab.

Solving → Run Calculation

	Run Calculation	
Update Dynamic Mesh	No (The - 1000 -	1.
Input Summary	No. of Iterations 1000	-/
Advanced	🗸 Check Case	Calculate

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

The solution converges within approximately 55 iterations.

Throughout the calculation, Fluent displays a warning in the console regarding reversed flow at the outlets. This behavior is expected in this case since air is redirected to the outlets, creating small regions of recirculation.

Note

The warning message can be switched off by setting the <code>solve/set/flow-warnings</code> text user interface (TUI) command to no in the console.

4. Close Fluent.

File → Close Fluent

5. Save the project in ANSYS Workbench.

$\textbf{File} \rightarrow \textbf{Save}$

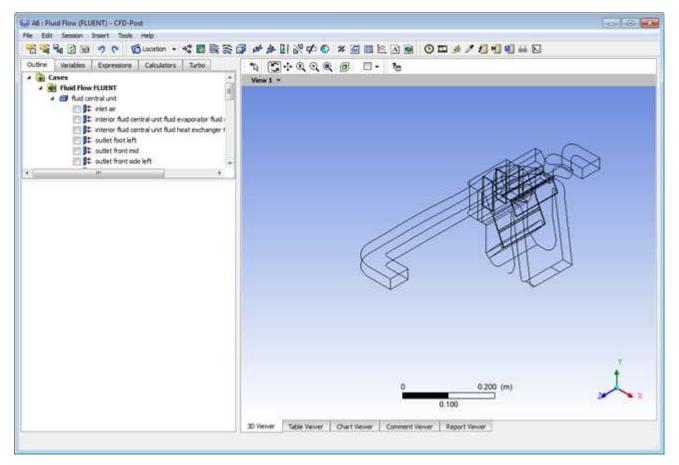
2.4.6. Postprocessing and Setting the Output Parameters in ANSYS CFD-Post

In this step, you will visualize the results of your CFD simulation using ANSYS CFD-Post. You will plot vectors that are colored by pressure, velocity, and temperature, on a plane within the geometry. In addition, you will create output parameters within ANSYS CFD-Post for later use in ANSYS Workbench.

In the ANSYS Workbench **Project Schematic**, double-click the **Results** cell in the ANSYS Fluent fluid flow analysis system to start CFD-Post. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.

The CFD-Post application appears with the automotive HVAC geometry already loaded and displayed in outline mode. Note that ANSYS Fluent results (that is, the case and data files) are also automatically loaded into CFD-Post.





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

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

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

🛛 🔽 Object Hig	hlighting	
Туре	Surface Mesh	
Pre-genera	ate region highlight	
Background		
Mode	Color	
Color Type	Solid	
Color]
ANSYS Logo	White	
Text Color		
Edge Color		
Axis Visibility 🛛 🖉 Ruler Visibilit		
Stereo		
Mode	Normal	
Stereo Effect	Weaker	
For stereo viev	reo Mode setting only takes effect the next time you run the application. wing to work, you need to turn on 'Stereo' in your graphic card, have a display that supports stereo, and ensure is set to Perspective mode (Right-click in viewer > Projection > Perspective).	e

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

Tip

You can also click the ellipsis icon which you can choose an arbitrary color.

- d. Click **OK** to set the white background color for the display and close the **Options** dialog box.
- 2. Adjust the color-map legend to show the numbers in floating format.
 - a. In the **Outline** tree view, double-click **Default Legend View 1** to display the **Details** view for the default legend to be used for your plots.
 - b. In the **Definition** tab of the **Details** view, from the **Title Mode** drop-down list, select **Variable**.

Details of Default Legend View 1

Definition 4	ppearance	
Title Mode Show Lege	Variable 🔹	
Vertical	Horizontal	
Location		
X Justification	Left 🔹	
Y Justification	Тор 💌	
Position	0.02 0.15	

c. In the Appearance tab, set the Precision to 2 and Fixed.

Definition	Appearance
Sizing Para	meters
Size	0.6
Aspect	0.07
- Text Param	eters
Precision	2 Fixed -
Value Ticks	5
Font	Sans Serif 👻
Color Mode	Default
Colour	
Text Rotatio	n O
Text Height	0.024

Details of Default Legend View 1

- d. Click **Apply** to set the display.
- 3. Plot vectors colored by pressure.
 - a. From the main menu, select **Insert** \rightarrow **Vector** or click $\stackrel{\clubsuit}{\Rightarrow}$ in the ANSYS CFD-Post toolbar.

This displays the **Insert Vector** dialog box.

- b. Keep the default name of **Vector 1** by clicking **OK**.
- c. In the **Details** view for **Vector 1**, under the **Geometry** tab, configure the following settings.

Details of Vecto	r 1						
Geometry	Color	lor Symbol Render View					
Domains							
Definition							
Locations	symm	netry central	unit		•		
Sampling	Equa	Equally Spaced 👻					
# of Points	1000	0			*		
Variable	Veloc	ity			•		
Boundary Data	Э	🔘 Hybrid		Ocn:	servative		
Projection	Tang	ential			•		

- i. Ensure All Domains is selected from the Domains drop-down list.
- ii. From the Locations drop-down list, select symmetry central unit.
- iii. From the **Sampling** drop-down list, select **Equally Spaced**.
- iv. Set the **# of Points** to 10000.
- v. From the **Projection** drop-down list, select **Tangential**.
- d. In the **Color** tab, configure the following settings.

Geometry	Color	Symbol	Render	View		
Mode	Variable	e			•	
Variable	Press	ure			•)
Range	Globa				•	
Min				u	nknown	
Max				u	nknown	
Boundary Dat	a	🔘 Hybrid		One Construction	servative	
Color Scale	Linear	,			•	
Color Map	Defau	ılt (Rainbow)		•	ł
Undef. Color						

Details of **Vector 1**

i. From the **Mode** drop-down list, select **Variable**.

- ii. From the Variable drop-down list, select Pressure.
- e. In the **Symbol** tab, configure the following settings.

Details of **Vector 1**

Geometry	Color	Symbol	Render	View	
Symbol	Line Ar	row			•
Symbol Size	0.05				
🔽 Normalize	9 Symbols				

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

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

In the graphics display window, note that **symmetry-central-unit** shows the vectors colored by pressure. Use the controls in CFD-Post to rotate the geometry (for example, clicking the dark blue axis in the axis triad of the graphics window). Zoom into the view as shown in Figure 2.16: Vectors Colored by Pressure (p. 105).

Note

To better visualize the vector display, you can deselect the **Wireframe** view option under **User Locations and Plots** in the **Outline** tree view.

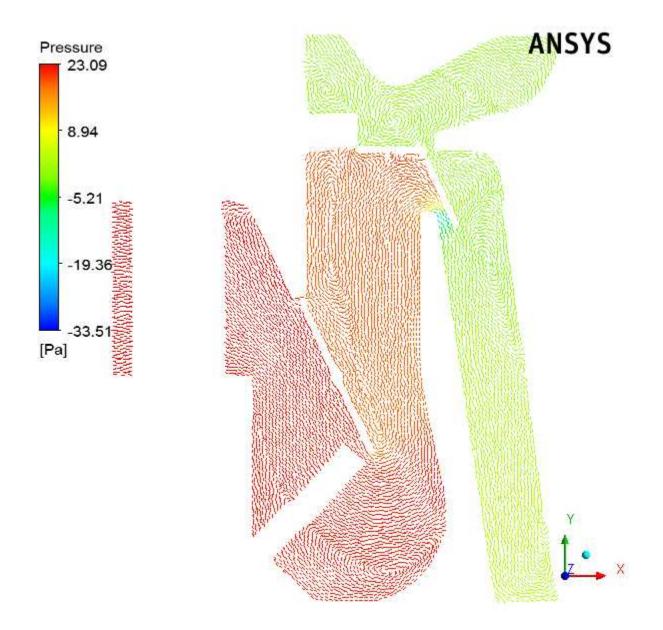
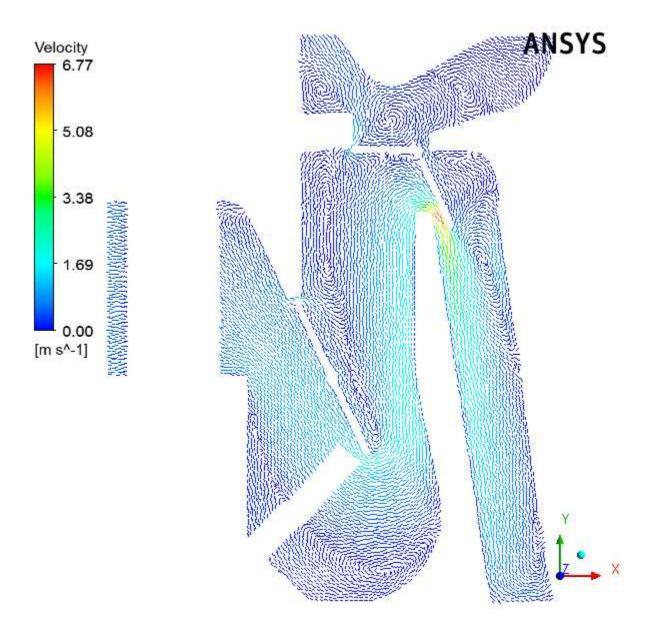


Figure 2.16: Vectors Colored by Pressure

- 4. Plot vectors colored by velocity.
 - a. In the **Details** view for **Vector 1**, under the **Color** tab, configure the following settings.
 - i. Select **Velocity** from the **Variable** drop-down list.
 - ii. Click **Apply**.

The velocity vector plot appears on the **symmetry-central-unit** symmetry plane.





- 5. Plot vectors colored by temperature.
 - a. In the **Details** view for **Vector 1**, under the **Color** tab, configure the following settings.
 - i. Select Temperature from the Variable drop-down list.
 - ii. Select User Specified from the Range drop-down list.
 - iii. Enter 273 K for the **Min** temperature value.
 - iv. Enter 310 K for the **Max** temperature value.
 - v. Click Apply.

The user-specified range is selected much narrower than the Global and Local ranges in order to better show the variation.

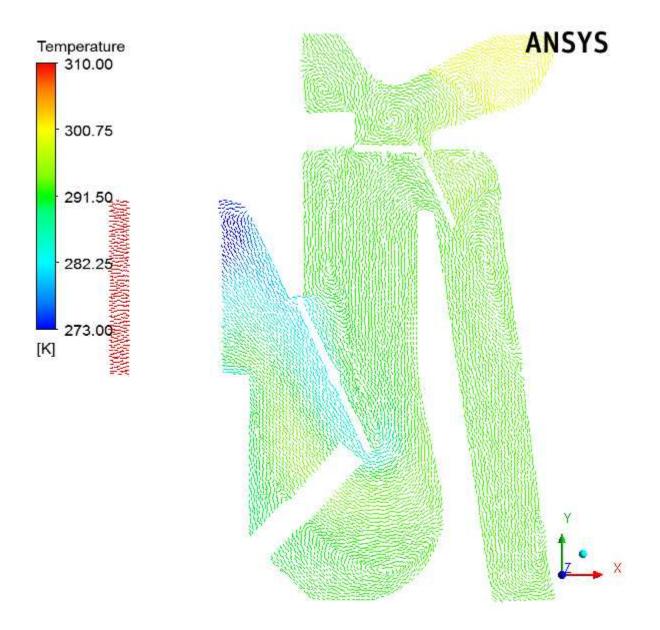


Figure 2.18: Vectors Colored by Temperature

Note the orientation of the various valves and how they impact the flow field. Later in this tutorial, you will change these valve angles to see how the flow field changes.

6. Create two surface groups.

Surface groups are collections of surface locations in CFD-Post. In this tutorial, two surface groups are created in CFD-Post that will represent all of the outlets and all of the front outlets. Once created, specific commands (or expressions) will be applied to these groups in order to calculate a particular numerical value at that surface.

- a. Create a surface group consisting of all outlets.
 - i. With the **Outline** tree view open in the CFD-Post tree view, open the **Insert Surface Group** dialog box.

Insert \rightarrow Location \rightarrow Surface Group

💿 Ins	ert Surface Group 📑 💌
Name	alloutlets
	DK Cancel

- ii. Enter alloutlets for the Name of the surface group, and click OK to close the Insert Surface Group dialog box.
- iii. In the **Details** view for the alloutlets surface group, in the **Geometry** tab, click the ellipsis icon

mext to **Locations** to display the **Location Selector** dialog box.

Location Selector
▲ Fluid Flow FLUENT
Ĵ‡ inlet air
Ĵ‡ interior fluid central unit fluid evaporator fluid central unit
Ĵ‡ interior fluid central unit fluid evaporator fluid evaporator
Ĵ‡ interior fluid central unit fluid heat exchanger fluid central unit
🕽 🗱 interior fluid central unit fluid heat exchanger fluid heat exchanger
Ĵ‡ outlet foot left
Ĵ‡ outlet front mid
Ĵ‡ outlet front side left
Ĵ‡ outlet windshield
🕽 🗱 symmetry central unit
Ĵ‡ symmetry evaporator
🕽 🗱 symmetry heat exchanger
)‡ wall central unit
)‡ wall evaporator
Ĵ‡ wall ftpos
)‡ wall hcpos
)‡ wall heat exchanger
)‡ wall inlet flow diverter
Ĵ‡ wall wsfpos
OK <u>C</u> ancel

- iv. Select all of the outlet surfaces (outlet foot left, outlet front mid, outlet front side left, and outlet windshield) in the Location Selector dialog box (hold Ctrl for multiple selection) and click OK.
- v. Click **Apply** in the **Details** view for the new surface group.

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

b. Create a surface group for the front outlets.

Perform the same steps as described above to create a surface group called frontoutlets with locations for the front outlets (outlet front mid and outlet front side left).

7. Create expressions in CFD-Post and mark them as ANSYS Workbench output parameters.

In this tutorial, programmatic commands or expressions are written to obtain numerical values for the mass flow rate from all outlets, as well as at the front outlets, windshield, and foot outlets. The surface groups you defined earlier are used to write the expressions.

- a. Create an expression for the mass flow from all outlets.
 - i. With the **Expressions** tab open in the CFD-Post tree view, open the **Insert Expression** dialog box.

Insert → **Expression**

.

ii. Enter floutfront for the Name of the expression and click OK to close the Insert Expression dialog box.

Details of flou	ittront		
Definition	Plot	Evaluate	
-(massFlow	v()@fror	toutlets)*2	2
Value		-0.000264	112 [kg s^-1]
Apply			Reset

iii. In the **Details** view for the new expression, enter the following in the **Definition** tab.

-(massFlow()@frontoutlets)*2

The sign convention for **massFlow()** is such that a positive value represents flow into the domain and a negative value represents flow out of the domain. Since you are defining an expression for outflow from the ducts, you use the negative of the **massFlow()** result in the definition of the expression.

iv. Click **Apply** to obtain a **Value** for the expression.

Note the new addition in the list of expressions in the **Expressions** tab in CFD-Post.

In this case, there is a small net backflow into the front ducts.

- v. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu. A small "P" with a right-pointing arrow appears on the expression's icon.
- b. Create an expression for the mass flow from the wind shield.
 - i. Perform the same steps as described above to create an expression called floutwindshield with the following definition:

-(massFlow()@outlet windshield)*2

- ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.
- c. Create an expression for the mass flow from the foot outlets.
 - i. Perform the same steps as described above to create an expression called floutfoot with the following definition:

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

- ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.
- d. Create an expression for the mass weighted average outlet temperature.
 - i. Perform the same steps as described above to create an expression called outlettemp with the following definition:

massFlowAveAbs(Temperature)@alloutlets

- ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.
- 8. Close ANSYS CFD-Post.

In the main menu, select **File** \rightarrow **Close CFD-POST** to return to ANSYS Workbench.

- 9. In the **Outline of All Parameters** view of the **Parameter Set** tab (double-click **Parameter Set**), review the newly-added output parameters that you specified in ANSYS CFD-Post and when finished, click the **Project** tab to return to the **Project Schematic**.
- 10. If any of the cells in the analysis system require attention, update the project by clicking the **Update Project** button in the ANSYS Workbench toolbar.
- 11. Optionally, review the list of files generated by ANSYS Workbench. If the **Files** view is not open, select **View** → **Files** from the main menu.

You will notice additional files associated with the latest solution as well as those generated by CFD-Post.

File View Tools Units Extensions 3					
🗋 🎯 🛃 🔣 🦳 Project 🕼 Param	ieter set	×			
Dimport	ct 🍠 U	pdate Project 💔 Update All Design Points 📲	ACT Start P	age	
		Schematic			
Analysis Systems					
Fluid Flow (Fluent)					
IC Engine (Fluent)		• A			
Throughflow		1 S Flud Flow (FLUENT)			
Throughflow (BladeGen)		2 🕼 Geometry			
E Component Systems		3 🍘 Mesh 🧹			
E Design Exploration					
Edemal Connection Systems		4 🔐 Setup 🗸 🖌			
		5 Solution			
		6 🥪 Results 🗸 🖌			
		7 D Parameters			
		Fluid Flow (FLUENT)			
		Find Flow (FLOENT)			
		Plate now (PLOENT)			
		Full Flow (FLOENT)			
	63				
		Parameter Set			
	්දා Files				
			8	c	D
	Files	Parameter Set	8 Ce *	Sze 💌	Туре
	Files	Parameter Set	Ce •	Size • 235 KB	Type Workbench Projec
	Files 1 2 3	Parameter Set A Name A Name Geom, agdb	Ce *	Size Contract Contra	Type Workbench Project Geometry Fie
	Files 1 2 3 4	Parameter Set A Name A Name Geom.agdb FFF.agdb	Ce • A2 A2	Size	Type Workbench Projec Geometry File Geometry File
	Files 1 2 3 4 5	Parameter Set A Name Filent-workbench-param.wbpj Geom.agdb FFF.agdb FFF.mshdb	Ce • A2 A2 A3	Size	Type Workbench Project Geometry File Jmshdb
	Files 1 2 3 4 5 6	Parameter Set	Ce • A2 A2 A3 A3,A4	Size	Type Workbench Project Geometry File Jimshdb Filuent Mesh File
	Files 1 2 3 4 5 6 7	Parameter Set	Ce • A2 A2 A3 A3,A4 A4	Size * 235 KB 2 MB 2 MB 28 MB 73 MB 217 KB	Type Workbench Project Geometry File .mshdb Filuent Mesh File FLUENT Model File
	Files 1 2 3 4 5 6	Parameter Set A Name fuent-workbench-param.wbp) Geom.agdb FFF.agdb FFF.mshdb FFF.msh FFF.set FFF.set FFF.l.cas.gz	Ce	Size 235 KB 2 MB 2 MB 2 MB 28 MB 73 MB 217 KB 30 MB	Type Workbench Project Geometry File Geometry File Josef Pluent Mesh File Fluent Mesh File FLUENT Model File FLUENT Case File
	Files 1 2 3 4 5 6 7	Parameter Set A Name fluent-workbench-param.wbp) Geom.agdb FFF.agdb FFF.mshdb FFF.msh FFF.set FFF.set FFF.set FFF.1.cas.gz FFF-1.cos.gz FFF-1.00055.dat.gz	Ce • A2 A2 A3 A3,A4 A4	Size * 235 KB 2 MB 2 MB 28 MB 73 MB 217 KB	Type Warkbench Project Geometry File Jinshdb Fluent Mesh File FLUENT Model File FLUENT Case File FLUENT Data File
	1 2 3 4 5 6 7 8	Parameter Set A Name fuent-workbench-param.wbp) Geom.agdb FFF.agdb FFF.mshdb FFF.msh FFF.set FFF.set FFF.l.cas.gz	Ce	Size 235 KB 2 MB 2 MB 2 MB 28 MB 73 MB 217 KB 30 MB	Type Workbench Project Geometry File Geometry File Josef Pluent Mesh File Fluent Mesh File FLUENT Model File FLUENT Case File
	FICS 1 2 3 4 5 6 7 8 9	Parameter Set A Name fluent-workbench-param.wbp) Geom.agdb FFF.agdb FFF.mshdb FFF.msh FFF.set FFF.set FFF.set FFF.1.cas.gz FFF-1.cos.gz FFF-1.00055.dat.gz	Ce	Size ▼ 235 KB 2 MB 2 MB 28 MB 73 MB 217 KB 30 MB 36 MB	Type Warkbench Project Geometry File Jinshdb Fluent Mesh File FLUENT Model File FLUENT Case File FLUENT Data File
Yiew All / Customize	Files 1 2 3 4 5 6 7 8 9 10	Parameter Set A Name fluent-workbench-param.wbp) Geom.agdb FFF.agdb FFF.agdb FFF.msh FFF.set FFF.set FFF.1.cas.gz FFF-1.cos.gz FFF-1.do055.dat.gz FFF.1.do055.dat.gz FFLuent.cst	Ce	Size ▼ 235 KB 2 MB 2 MB 2 MB 73 MB 73 MB 217 KB 30 MB 36 MB 48 KB	Type Workbench Project Geometry File Jinshdb Fluent Mesh File FLUENT Model File FLUENT Case File FLUENT Data File CFD-Post State Fil

Figure 2.19: The Updated Project Loaded into ANSYS Workbench Displaying the Files View

12. Save the project in ANSYS Workbench.

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

Note

You can also select the Save Project option from the CFD-Post File ribbon tab.

2.4.7. Creating Additional Design Points in ANSYS Workbench

Parameters and design points are tools that allow you to analyze and explore a project by giving you the ability to run optimization and what-if scenarios. Design points are based on sets of parameter values. When you define input and output parameters in your ANSYS Workbench project, you are essentially working with a design point. To perform optimization and what-if scenarios, you create multiple

design points based on your original project. In this step, you will create additional design points for your project where you will be able to perform a comparison of your results by manipulating input parameters (such as the angles of the various valves within the automotive HVAC geometry). ANSYS Workbench provides a Table of Design Points to make creating and manipulating design points more convenient.

- 1. Open the Table of Design Points.
 - a. In the Project Schematic, double-click the **Parameter Set** bus bar to open the Table of Design Points view. If the table is not visible, select **Table** from the **View** menu in ANSYS Workbench.

View → Table

The table of design points initially contains the current project as a design point (DP0), along with its corresponding input and output parameter values.

Figure 2.20: Table of Design Points (with DP0)

	A	Ð	C	D	E	F	G	н	1
1	Name 💌	P1-htpos 💌	P2 - ftpos 💌	P3 - wsfpos 💌	P4 - input_htpps -	P5-mout_fipos -	P6 - input_ws/pos 💌	P7-in_velocity ·	P8+n_temp *
2	Units							ms^-1	к 🗵
3	DP 0 (Current)	25	25	90	15	25	90	0.5	310

From this table, you can create new design points (or duplicate existing design points) and edit them (by varying one or more input parameters) to create separate analyses for future comparison of data.

- 2. Create a design point (DP1) by duplicating the current design point (DP0).
 - a. Right-click the **Current** design point and select **Duplicate Design Point** from the context menu.

The cells autofill with the values from the **Current** row.

b. Scroll over to the far right to expose the **Retain** column in the table of design points, and ensure the check box in the row for the duplicated design point **DP 1** (cell N4) is selected.

This allows the data from this new design point to be saved before it is exported for future analysis.

- 3. Create another design point (DP2) by duplicating the DP1 design point.
 - a. Right-click the **DP1** design point and select **Duplicate Design Point** from the context menu.

Since this is a duplicate of DP1, this design point will also have its data retained.

4. Edit values for the input parameters for DP1 and DP2.

For DP1 and DP2, edit the values for your input parameters within the Table of Design Points as follows:

	input_hcpos	input_ftpos	input_wsfpos	in_velocity	in_temp
DP1	45	45	45	0.6	300
DP2	90	60	15	0.7	290

	A	Ð	C	D	E	F	G	н	1	
1.	Nane 💌	P1-htpos 💌	P2 - Rpos 🔹	P3 - xisfpos 💌	P4 - input_hopes ·	P5-mput_fipos	P6 - input_wsfpos	P7-m_velocity *	P8-m_temp	
2	Units							ms^-1	к <u>в</u>	
3	DP 0 (Current)	25	25	90	15	25	90	0.5	310	
4	DP 1	45	45	45	45	45	45	0.6	300	
5	DP 2	90	60	15	90	60	15	0.7	290	

Figure 2.21: Table of Design Points (with DP0, DP1, and DP2 Defined)

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

5. Update all of your design points.

Click the **Update all Design Points** button in the ANSYS Workbench toolbar. Alternatively, you can also select one or more design points, right-click, and select **Update Selected Design Points** from the context menu. Click **OK** to acknowledge the information message notifying you that some open editors may close during the update process. By updating the design points, ANSYS Workbench takes the new values of the input parameters for each design point and updates the components of the associated system (for example, the geometry, mesh, settings, solution, and results), as well as any output parameters that have been defined.

Note

It may take significant time and/or computing resources to re-run the simulations for each design point.

6. Export the design points to separate projects.

This will allow you to work with calculated data for each design point.

- a. Select the three design points, DP0, DP1, and DP2 (hold Shift for multiple selection).
- b. Right-click the selected design points and select **Export Selected Design Points**.

Note the addition of three more ANSYS Workbench project files (and their corresponding folders) in your current working directory (fluent-workbench-param_dp0.wbpj, fluent-workbench-param_dp1.wbpj and fluent-workbench-param_dp2.wbpj). You can open each of these projects up separately and examine the results of each parameterized simulation.

Tip

You can easily access files in your project directory directly from the **Files** view by right-clicking any cell in the corresponding row and selecting **Open Containing Folder** from the menu that opens.

7. Inspect the output parameter values in ANSYS Workbench.

Once all design points have been updated, you can use the table of design points to inspect the values of the output parameters you created in CFD-Post (for example, the mass flow parameters at the various outlets: floutfront, floutfoot, floutwindshield, and outlettemp). These, and the rest of the output parameters are listed to the far right in the table of design points.

Figure 2.22: Table of Design Points (Showing Output Parameters for DP0, DP1, and DP2)

	A	E	Ŧ.	G	н	1	3	ĸ	L	M
1	Name 💌	P4- nout_hopes	PS- nout_fibes	PE - input_visifpos	P7 - in_velocity	PE- in_benp	P9 - floutfront	P10 - foubrindshield	P11- flautfoot	P12- outlettemp
2	Units				m s^-1	к 💌	kg s^-1	kgs^-1	kg s^+1	ĸ
3	DP 0 (Current)	15	25	90	0.5	310	-0.00026376	0.0011285	0.01016	292.34
4	OP 1	45	45	45	0.6	300	9.5785E-05	0.0071258	0.0060084	284.98
5	DP 2	90	60	15	0.7	290	0.0016708	0.0081473	0.0055169	277.13
						100010				

- 8. Click the **Project** tab, just above the ANSYS Workbench toolbar to return to the **Project Schematic**.
- 9. View the list of files generated by ANSYS Workbench (optional).

$\textbf{View} \rightarrow \textbf{Files}$

The additional files for the new design points are stored with their respective project files since you exported them.

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

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

11. Quit ANSYS Workbench.

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

2.4.8. Postprocessing the New Design Points in CFD-Post

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

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

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

- b. Open CFD-Post by double-clicking the **Results** cell in the Project Schematic for the Fluid Flow (Fluent) analysis system.
- c. View the vector plot colored by temperature. Ensure that **Range** in the **Color** tab is set to **User Specified** and the **Min** and **Max** temperature values are set to 273 K and 310 K, respectively.

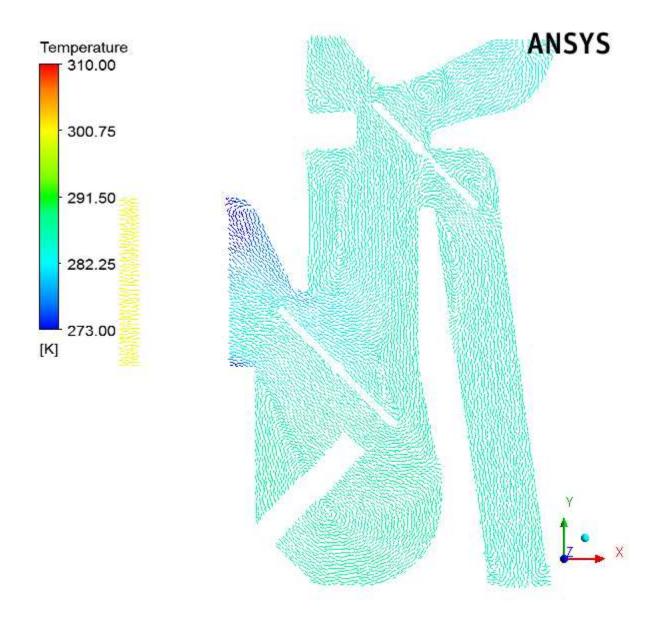
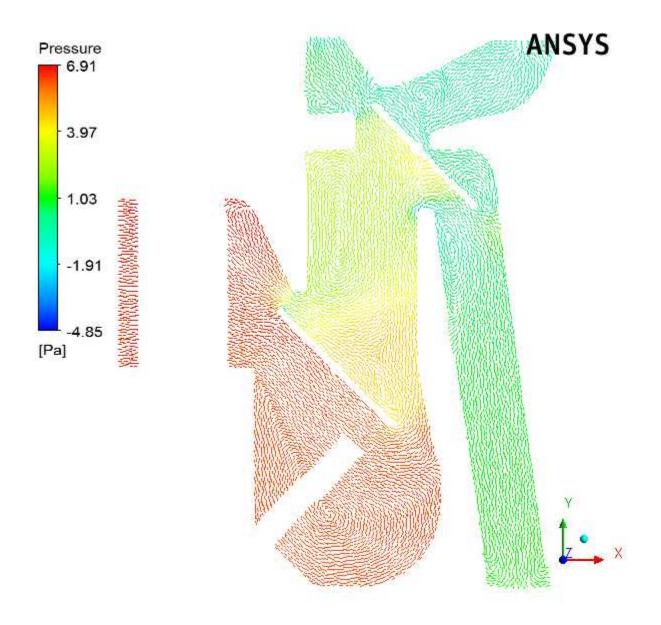


Figure 2.23: Vectors Colored by Temperature (DP1)

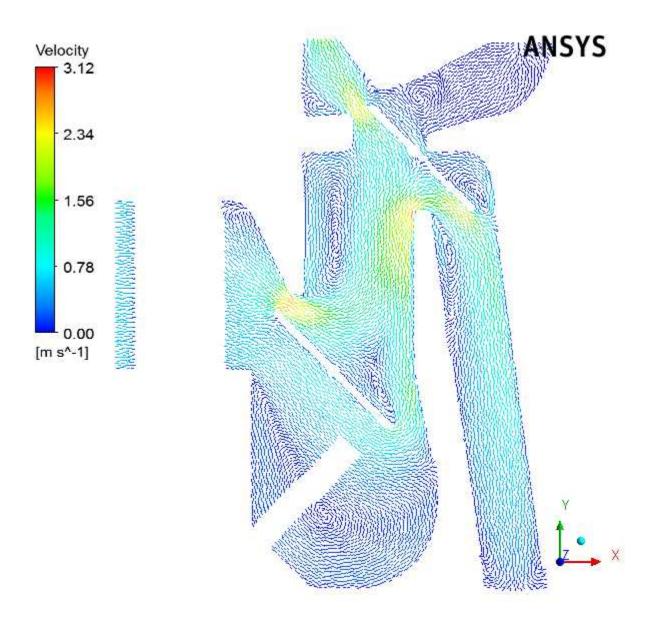
d. View the vector plot colored by pressure. Ensure that **Range** in the **Color** tab is set to **Global**.





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





- f. When you are finished viewing results of the design point DP1 in ANSYS CFD-Post, select **File** → **Close CFD-Post** to quit ANSYS CFD-Post and return to the ANSYS Workbench **Project Schematic**, and then select **File** → **Exit** to exit from ANSYS Workbench.
- 2. Study the results of the second design point (DP2).
 - a. Open the ANSYS Workbench project for the second design point (DP2).

In your current working folder, double-click the fluent-workbench-param_dp2.wbpj file to open ANSYS Workbench.

b. Open CFD-Post by double-clicking the **Results** cell in the Project Schematic for the Fluid Flow (Fluent) analysis system.

c. View the vector plot colored by temperature. Ensure that **Range** in the **Color** tab is set to **User Specified** and the **Min** and **Max** temperature values are set and the **Min** and **Max** temperature values are set to 273 K and 310 K, respectively.

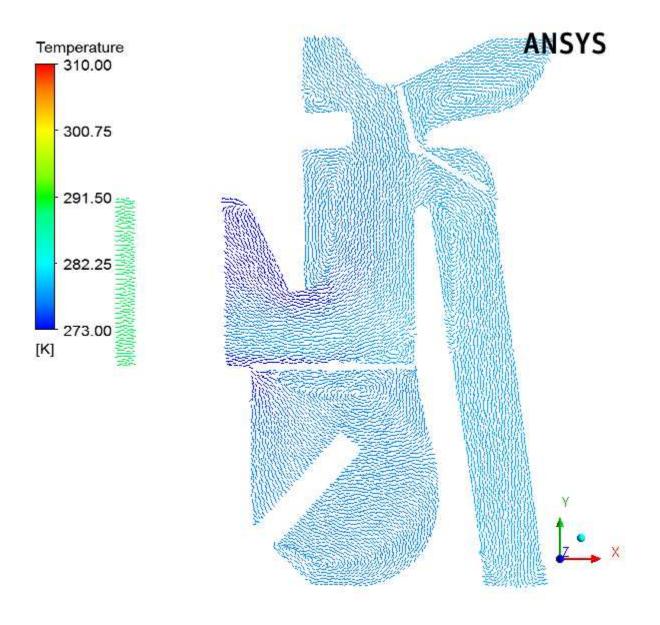


Figure 2.26: Vectors Colored by Temperature (DP2)

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

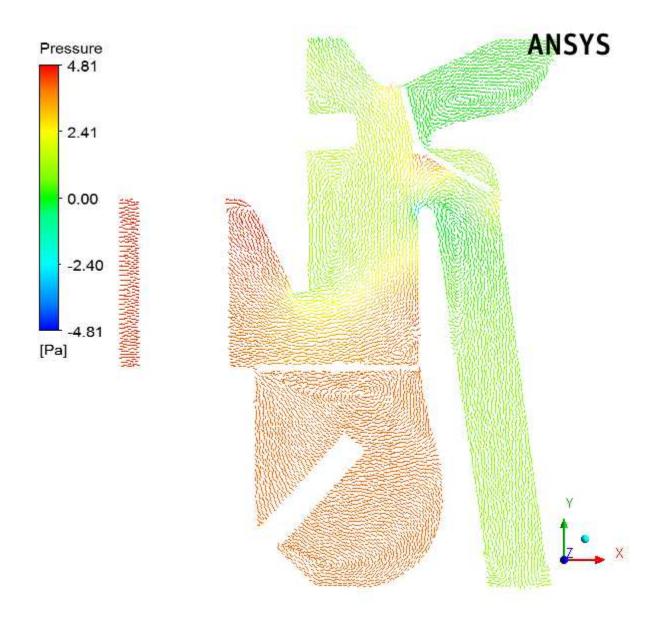
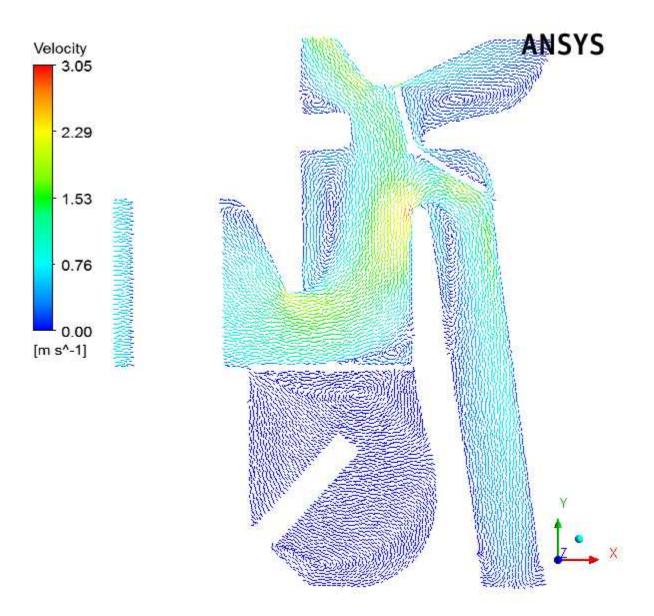


Figure 2.27: Vectors Colored by Pressure (DP2)

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





3. When you are finished viewing results in ANSYS CFD-Post, select **File** → **Close CFD-Post** to quit ANSYS CFD-Post and return to the ANSYS Workbench **Project Schematic**, and then select **File** → **Exit** to exit from ANSYS Workbench.

2.4.9. Summary

In this tutorial, input and output parameters were created within ANSYS Workbench, ANSYS Fluent, and ANSYS CFD-Post in order to study the airflow in an automotive HVAC system. ANSYS Fluent was used to calculate the fluid flow throughout the geometry using the computational mesh, and ANSYS CFD-Post was used to analyze the results. ANSYS Workbench was used to create additional design points based on the original settings, and the corresponding simulations were run to create separate projects where parameterized analysis could be performed to study the effects of variable angles of the inlet valves, velocities, and temperatures. Also, note that simplified solution settings were used in this tutorial to speed up the solution time. For more improved solution accuracy, you would typically use denser mesh and higher order discretization for all flow equations.