To use ANSYS Fluent in your house, please use VDI (See below Link) https://etc.engineering.uiowa.edu/help-desk/how-use/vdi-how-use-virtual-windows-desktop

Simulation of Turbulent Flow in an Asymmetric Diffuser

ME:5160 Intermediate Mechanics of Fluids CFD LAB 3 (ANSYS 2022 R1; Last Updated: July 19, 2022)

By Timur Dogan, Michael Conger, Dong-Hwan Kim, Sung-Tek Park, Christian Milano, Maysam Mousaviraad, Tao Xing and Fred Stern

> IIHR-Hydroscience & Engineering The University of Iowa C. Maxwell Stanley Hydraulics Laboratory Iowa City, IA 52242-1585

1. Purpose

The Purpose of CFD Lab 3 is to simulate **turbulent** flows inside a diffuser following the "CFD process" by an interactive step-by-step approach and conduct verifications. Students will have "hands-on" experiences using ANSYS to conduct **validation of velocity, turbulent kinetic energy, and skin friction factor. Effect of turbulent models will be investigated, with/without separations**. Students will manually generate meshes, solve the problem and use post-processing tools (contours, velocity vectors, and streamlines) to visualize the flow field. Students will analyze the differences between CFD and EFD and present results in a CFD Lab report.



Flow Chart for "CFD Process" for diffuser

2. Simulation Design

The problem to be solved is that of turbulent flows inside an asymmetric diffuser (2D). Reynolds number is 17,000 based on inlet velocity and inlet dimension (D1). The following figure shows what the geometry looks like with definitions for all geometry parameters. Before the diffuser, a straight channel was used for generating fully developed channel flow at the diffuser inlet. You will conduct simulation for two different half angles of 4 and 10 with two different turbulence models of SST and k- ϵ .

Parameter	Symbol	Unit	Value
Inlet dimension	D1	m	2
Inlet length	L1	m	60
Diffuser half angle	α	degree	4 or 10
Outlet dimension	D2	m	9.4
Outlet length	L2	m	70

Table 1 – Main particulars



In CFD Lab3, all EFD data for turbulent airfoil flow in this Lab can be found on the class website <u>http://www.engineering.uiowa.edu/~me_160/</u>.

3. Starting with ANSYS Workbench

3.1. Create the layout as per below.



3.2 File > Save. Save the project on the network drive and Call it "*CFD Lab 3*".

4. Geometry Creation

In this section, we will create the geometry for the diffuser with 10 degree half angle, then copy and modify the geometry for the 4 degree half angled diffuser.

4.1 Right click Geometry and select New DesignModeler Geometry...



4.2 Make sure that Unit is set to Meter and Degree (default value).



4.3 Select XYplane and click New Sketch button.



4.4 Right click **Sketch1** and select **Look at**.

Tree Outline	ф.
⊡, 👰 A: 10 degree half angle	
🚊 🛶 🖈 XYPlane	
Sketch1	
ZXPlane 💡 Always Show Sketch	
YZPlane 🔮 Hide Sketch	
👘 0 Parts, 0 B 🕫 Look at	
Show Dependencies	
🗙 Delete	
🔣 Generate (F5)	
allo Rename (F2)	

4.5 **Sketching** > **Constraints** > **Auto Constraints**. Enable the auto constraints option to pick the exact point as below.

File Create Con	cept Tools Units View	Help	
🔄 📑 📑 🚳	Undo @Redo	Select: 🔭 🖏	- K K 🖪 🔳
■ • ■ • ⁄~ •	h+ k+ k+ k+ ;	🗶 🗹	
XYPlane 🔻	⊁ Sketch1 ▼	📁 📔 誟 Genera	te 🛛 🖤 Share Topology
Thin/Surface	🗣 Blend 🔻 🔦 Chamfer	Slice 🛛 🚸	Point 💽 Conversion
BladeEditor: 💒 Im	port BGD 🛛 🔏 Load BGD	Icad NDF	🖹 FlowPath 🛛 🥑 Blade
Ø ₩ 8 ≣	(
Sketching Toolboxes			ф.
	Draw	,	
	Modit	fy	
	Dimensi	ions	
	Constrain	ts	A
777 Fixed			
, H orizontal			
Vertical			
Perpendicular			
A Tangent			
✓Coincident			
Midpoint			
• î • Symmetry			
✓ Parallel			
Concentric			
A Equal Radius			
💒 Equal Length			
Equal Distance			
CON Auto Constraint	5		Global: _ Cursor: ✓
	Settings		~
Sketching Modelin	g		
Details View			ф.
Details of Sketch1			
Sketch	Sketch1		
Sketch Visibility	Show Sketch		
Show Constraints	' No		

4.6 Sketching > Draw > Line. Draw a vertical line on the y-axis starting from the origin as shown below (P indicates that the origin point is selected and V indicates that the line is vertical).





4.7 Sketching > Dimensions > General. Click on the vertical line then click on the left side of the line to place the dimension. Change the dimension in Details View to 2m (skip the unit ([m]) when put in the value).



4.8 **Sketching** > **Draw** > **Line**. Create a horizontal line on the x-axis starting at the origin as per below (**H** indicates that line is horizontal).



4.9 **Sketching** > **Dimensions** > **General**. Change the length of the horizontal line you created to 60m.



4.10 Sketching > Draw > Line. Create line at an angle with respect to x-axis as shown below.



4.11 Sketching > Dimensions > Angle. Select the line just created then select the x-axis then change the angle to 10°. (Note: if ANSYS gives a default exterior angle instead of the interior angle, right click and select Alternate Angle.)





4.12 Sketching > Draw > Line. Create a horizontal line as per below.

4.13 **Sketching** > **Dimensions** > **General**. Change the length of the line just created to 70m.



4.14 Sketching > Draw > Line. Draw the horizontal line circled in red line as per below.



4.15 Sketching > Constraints > Equal Length. Select two lines circled in red as shown below.

Sketching Toolboxes	4
Draw	
Modify	
Dimensions	
Constraints	A
Fixed Fixed Fixed If Vertical Vergendicular A Tangent Coincident ···· Midpoint A'symmetry VParallel @ Concentric X Found Radius & Equal Length & &	
Settings	
Sketching	





4.16 Sketching > Draw > Line. Draw the horizontal line circled in red as per below.

4.17 **Sketching** > **Constraints** > **Equal Distance**. Click on Point 1 and then click on the Point 2. Click Point 3 and then click on line 4. This makes points 1 and 3 the same distance from the y-axis in the horizontal direction.



4.18 Sketching > Draw > Line. Draw the horizontal line circled in red as shown below.



4.19 Sketching > Constraints > Equal Length. Click on two lines circled in red as below.



4.20 Sketching > Draw > Line. Draw the final line circled in red as shown below. When you draw this line, if all previous dimensions and constraints are correct, the line should have two P's at the ends with a V in the center. This indicates that the line starts and ends on the two points and is perfectly vertical. If you do not get the V, recheck all dimensions and constraints.

A10		\frown
	~	
	H11	U

4.21 **Sketching** > **Dimensions** > **General**. Change the length of the line circled in red to 9.4m, this will automatically adjust the length of the expansion region because of the applied constraints.



De	etails View	д
-	Details of Sketch1	
	Sketch	Sketch1
	Sketch Visibility	Show Sketch
	Show Constraints?	No
-	Dimensions: 5	
	A3	10 °
	H2	60 m
	H4	70 m
	🗌 V1	2 m
	V5	9.4 m
-	Edges: 8	
	Line	Ln7
	Line	Ln8
	Line	Ln9
	Line	Ln10
	Line	Ln11
	Line	Ln12
	Line	Ln13
	Line	Ln14

4.22 **Concept** > **Surfaces From Sketches**. Select the sketch you created and click **Apply** then click **Generate**. This will create a surface as shown below.



4.23 **Tools** > **Face Split**. Select the surface you created (it will be highlighted in green when you select it as shown below) then click **Apply** for **Target Face**.



4.24 Click on the yellow region shown below.

De	etails View	т
-	Details of Faces	Split1
	Face Split	FaceSplit1
Ξ	Face Split Grou	p 1 (RMB)
	Face Split Type	By Points and Edges
	Target Face	1
	Tool Geometry	0



4.25 While holding **Ctrl** button click on the two points circled in red then click **Apply** button.

4.26 Click on the region marked with red rectangle below.

Target Face

Tool Geometry

Face Split Type By Points and Edges

1

De	etails View	₽
-	Details of Faces	Split1
	Face Split	FaceSplit1
Ξ	Face Split Grou	p 1 (RMB)
	Face Split Type	By Points and Edges
	Target Face	1
	Tool Geometry	1

Cancel

Apply

4.27 While holding **Ctrl** button click on the two points circled in red then click **Apply** button.



File Create Concent Tools Units View Hal	In											
The create concept roots onto view her	ne Jaab 🕅 🕴			1.545.346	1 C A @ (n 150 #10 J	e 1 (2				
	seccila L _∠	։ ինդ և և			ાઝજર	य थ थ थ थ	≪	• • • • • • • • • • • • • • • • • • •				
		_	_			1						
XYPlane • 👫 Sketch1 • 🖉	3 Gene	rate Share	Topology 🛃	Parameter	s Extrude	Revolve L	.Sweep 🛛 🐥 Skin/	Loft				
] 🖿 Thin/Surface 💊 Blend 🔻 🦴 Chamfer 🐞	Slice	👂 Point 🔹 Co	nversion									
BladeEditor: 🎇 Import BGD / 🖓 Load BGD 🛷	Load NDF	式 FlowPath	🥩 Blade 🖪	🕈 Splitter	VistaTFExport	SuportPoints	StageFluidZo	ne 🛛 🛃 SectorCu	it 🌾 ThroatArea	💕 CAD Import 👻	Preferences	
遊巫 8 = (海邊	•			90	🗰 🧭							
Tree Outline	4	Graphics										4
Sketching Modeling				—H6——						H2		<u> </u>
Details View	4											
Details of FaceSplit1												
Face Split FaceSplit1												
Face Split Group 1 (RMB)												v
Face Split Type By Points and Edges												1
Target Face 1												t.
1001 Geometry 2												•
						0.00	30.0	45.00	60.00 (m			ĕ —→ ×

4.28 Click the Generate button and Save your progress.

4.29 Close the ANSYS Design Modeler and update geometry



4.30 Right click on geometry and select Duplicate.



4.31 Rename the new geometry file as per below.



4.32 Open the new geometry file you created and select Sketch1 under the tree outline as per below. Change the half angle to **4 degrees** under details view as per below then click the **Generate** button.

	Sketching Modeling
2 Seometry V 2 Setup ?	Details View
10 degree half angle Mesh 3 🕼 Solution 💡	Sketch Visibility Show Sketch
k-e	Show Constraints? No
	Dimensions: 5
	A3 4°
	H4 70 m
	H6 60 m
	□ V1 2 m
4 degree balt	□ V5 9.4 m
Edit Geometry in SpaceClaim	Edges: 8
Edit Geometry in Discovery	Line Ln7
Tree Outline	Line Ln9
□ → Jog D: 4 degree hair angle	Line Ln10
ZXPlane	Line Ln11
	Line Ln12
	Line Ln13
FaceSplit1	Line Ln14
i < 🕼 1 Part, 1 Body	Line Ln15

4.33 Save your file and quit ANSYS Design Modeler

5. Mesh Generation

This section shows how to generate the mesh for both 4 degree and 10 degree half angle cases.

5.1 Right click on Mesh and click Edit...



5.2 Right click on **Mesh** then select **Insert** > **Face Meshing**.



5.3 Select all three surface while holding Ctrl button and click Apply located Geometry.

-	Scope		
	Scoping Method	Geometry Selection	
	Geometry	Apply	Cancel
	Definition		
	Suppressed	No	
	Mapped Mesh	Yes	
	Method	Quadrilaterals	
	Internal Number of Divisions	Default	
	Constrain Boundary	No	

5.4 Select the **Edge** button. This will allow you to select edges of your geometry.



5.5 Right click on **Mesh** and **Insert > Sizing**.



0		(
D	etails of "Edge Sizing	g" - Sizing	Ļ	
E	Scope			
	Scoping Method	Geometry Selec	tion	
	Geometry	Apply	Cancel	
E	Definition			
	Suppressed	No		
	Туре	Element Size		
	Element Size	Default (2.3017	m)	
E	Advanced			
	Behavior	Soft		
	Growth Rate	Default (1.2)		
	Capture Curvature	Curvature No		
	Capture Proximity	No		
	Bias Type	No Bias		

5.7 Change parameter for Edge Sizing as per below (Left edge is shown as an example).

D	etails of "Edge Sizing" - S	iizing 🛛 🕈
E	Scope	
	Scoping Method	Geometry Selection
	Geometry	2 Edges
E	Definition	
	Suppressed	No
	Туре	Number of Divisions
	Number of Divisions	59
E	Advanced	
	Behavior	Hard
	Capture Curvature	No
	Capture Proximity	No
	Bias Type	
	Bias Option	Bias Factor
	Bias Factor	15.106

5.8 Right click on **Mesh** and **Insert** > **Sizing**.

5.9 While holding **Ctrl**, click on the edge shown below and click **Apply**.



5.10 Change parameter for **Edge Sizing** as per below and click **Apply** (Right edge is shown as an example).

De	etails of "Edge Sizing 2" -	Sizing 4
Ξ	Scope	
	Scoping Method	Geometry Selection
	Geometry	2 Edges
Ξ	Definition	
	Suppressed	No
	Туре	Number of Divisions
	Number of Divisions	59
	Advanced	
	Behavior	Hard
	Capture Curvature	No
	Capture Proximity	No
	Bias Type	
	Bias Option	Bias Factor
	Bias Factor	87.76

5.11 Right click on **Mesh** and **Insert** > **Sizing**.

5.12 While holding **Ctrl**, click on the edge shown below and click **Apply**.



5.13 Change parameter for Edge Sizing as per below and click Apply.

De	Details of "Edge Sizing 3" - Sizing					
	Scope					
	Scoping Method	Geometry Selection				
	Geometry	2 Edges				
	Definition					
	Suppressed	No				
	Туре	Number of Divisions				
	Number of Divisions	59				
	Advanced					
	Behavior	Hard				
	Capture Curvature	No				
	Capture Proximity	No				
	Bias Type					
	Bias Option	Bias Factor				
	Bias Factor	3.6776				
	Reverse Bias	No Selection				

5.14 Right click on **Mesh** and **Insert** > **Sizing**.



5.15 While holding **Ctrl** click on the edge shown below and click **Apply**.

5.16 Change parameter for Edge Sizing as per below and click Apply.

De	Details of "Edge Sizing 4" - Sizing					
	Scope					
	Scoping Method	Geometry Selection				
	Geometry	2 Edges				
E	Definition					
	Suppressed	No				
	Туре	Number of Divisions				
	Number of Divisions	59				
	Advanced					
	Behavior	Hard				
	Capture Curvature	No				
	Capture Proximity	No				
	Bias Type					
	Bias Option	Bias Factor				
	Bias Factor	1.8593				



5.17 Right click on **Mesh** and **Insert** > **Sizing**.

5.18 While holding **Ctrl** click on the edge shown below and click **Apply**.



5.19 Change parameter for Edge Sizing as per below and click Apply.

De	Details of "Edge Sizing 5" - Sizing			
Ξ	Scope			
	Scoping Method	Geometry Selection		
	Geometry	2 Edges		
Ξ	Definition			
	Suppressed	No		
	Туре	Number of Divisions		
	Number of Divisions	59		
	Advanced			
	Behavior	Hard		
	Capture Curvature	No		
	Capture Proximity	No		
	Bias Type			
	Bias Option	Bias Factor		
	Bias Factor	4.3763		
	Reverse Bias	No Selection		



5.20 Mesh > Physics Preference. Change from Mechanical to CFD (Once you click the Mesh under the Outline, detailed options will appear as below).



5.21 Click the Generate Mesh button.



- Outline Name Search Outline Project* 🖮 🐻 Mod<u>el (B2)</u> 🗙 🔄 Surface Body 🖉 🗖 Materials 🗄 🧹 🙀 Coordinate Systems ⊡….,∕© Mesh 🏹 🎇 Face Meshing √ 🕼 Edge Sizing 🏑 🕼 Edge Sizing 2 🏑 🕼 Edge Sizing 3 🏑 🕼 Edge Sizing 4 🏸 Edge Sizing 5 Automatio Mesh Display Selection Home ₽ <u>⊸</u>Cut × Delete Named Selection 💿 Images 🕶 🏷 Tags Selection Information 🔐 Report Preview 🖧 Mana **←→** m ft E X 🖪 Copy 🔍 Find 🔆 Coordinate System 🎁 Section Plane 🗟 Show Errors 🛛 🚟 Unit Converter 🕮 Key Assignments 💷 User D Generate 💭 Comment Units Worksheet Keyframe Animation Manage Views Print Preview Duplicate Paste La Tree * Full Screen CReset Annotation Outline Mesh Insert Took Lavout Outline ▼ # 🗆 × 🝳 🔍 📦 🗣 🖺 🙄 🔸 🍳 🍳 🍭 🍭 Select 💺 Mode= 🎊 🕞 🛅 🛅 🖷 🖷 🖷 🌾 🖤 👳 🧮 Clipboard= Name ▼ Search Outline 🗸 🗸 Edge (Ctrl+E) Face Meshing 10:30 AM Project* Select or highlight edges on your model. Use the Ctrl button or hold the mouse button to select multiple edges. Model (B2) ė Geometry Face Meshing (i) Press F1 for help. 🔆 🤆 Coordinate Systems 🐨 Mesh
- 5.22 Select Geometry to hide the mesh and click the Edge button.

5.23 While holding the Ctrl button select the three top edges and right click on them, then select Create Named Selection. Change the name to *top_wall* and click OK. Similarly name the *bottom_wall (bottom)*, *inlet (left)* and *outlet(right)*.



Selection Name	\times
top wall	×
Apply selected geometry	
 Apply geometry items of same: 	
Size	
Туре	
Location X	
Location Y	
Location Z	
Apply To Corresponding Mesh Nodes	
OK Cancel	

5.24 File > Save Project and quit ANSYS Mesh. Right click on Mesh and click Update

	_	Α			•		В		- C
ľ	🎾 Geor	netry			1	\$	Mesh		1 See Fluent
l	👂 Geor	netry	\checkmark		2	۲	Mesh		
10	0 degree	half a	nale				Mesh	¢,	Edit
-									Duplicate
									Transfer Data From New
									Transfer Data To New
-		D						1	Update
	<i>.</i>								Update Upstream Components
•	Geor	netry							Clear Generated Data
2	Geor	netry	\checkmark	4					D-C
4	deareet	alf ar	ale					\$	kettesn
			9.0						Reset

- 5.25 Repeat this process for 4 degree and 10 degree half angle cases.
- 5.26 You should have the project schematic below.



6. Setup

	А	
1	🥩 Geometry	
2	Geometry	< 4
	10 degree half a	ngle
	D	
▼	U	_
▼ 1	🥩 Geometry	
▼ 1 2	Geometry	 .
▼ 1 2	Geometry Geometry 4 degree half an	✓ ⊿ Igle
▼ 1 2	Geometry Geometry 4 degree half an	✓ ▲ Igle

6.1 Right click Setup and click Edit.

6.2 Check **Double Precision** and select **START**.

Sluent Launcher 2022 R1 (Setting Edit Only) -		×
Fluent Launcher	<mark>/\</mark> ns	sys
Simulate a wide range of steady and transient industrial application general-purpose setup, solve, and post-processing capabilities of A including advanced physics models for multiphase, combustion, ele and more.	ns using NSYS Fl ctrocher	the luent mistry,
Dimension		
(i) 2D		
⊖ 3D		
Options Double Precision Display Mesh After Read Do not show this panel a Load ACT	ing again	
Parallel (Local Machine)		
Solver Processes	1	\$
Solver GPGPUs per Machine	0	\$
 ✓ Show More Options ✓ Show Learning Resources Start Cancel Help 		



6.3 Tree > Setup > General > Mesh > Check. Set the parameters as per below.

6.4 Tree > Setup > Models > Viscous. Select parameters as per below and click OK(Apply).

📧 Viscous 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)	C2-Epsilon
 Transition k-kl-omega (3 eqn) 	1.92
 Transition SST (4 eqn) 	TKE Prandtl Number
O Reynolds Stress (5 eqn)	1
 Scale-Adaptive Simulation (SAS) 	TDR Prandtl Number
 Detached Eddy Simulation (DES) 	1.3
k-epsilon Model	
Standard	
🔿 Realizable	
Near-Wall Treatment	User-Defined Functions
Standard Wall Functions	Turbulent Viscosity
Scalable Wall Functions	none
O Non-Equilibrium Wall Functions	Prandtl Numbers
Enhanced Wall Treatment	TKE Prandtl Number
O Menter-Lechner	none
 User-Defined Wall Functions 	TDR Prandtl Number
Enhanced Wall Treatment Ontions	none
✓ Pressure Gradient Effects	
Options	
Curvature Correction	
Production Kato-Launder	
Production Limiter	
ОК Сап	cel Help

Name		Material Type		Order Materials by
air		fluid	•	 Name
Chemical Formula		Fluent Fluid Materials		O Chemical Formula
P	roperties Density [kn/m키	air Mixture none	•	Fluent Database GRANTA MDS Database User-Defined Database
Viscosity [kg/(m s		constant	•	Edit
	C	0.000147		
	C	nange/Create Delete Close Help		

6.5 Tree > Setup > Materials > Fluid > air. Change the fluid properties and then click Change/Create then click Close.

6.6 Tree > Setup > Boundary Conditions > Zone > inlet. Change parameters for inlet velocity. Use the table below for as per below and click OK(Apply).

🥌 Velocity	Inlet							×
Zone Name								
inlet								
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	Structure	UDS
٧	/elocity Spec	ification Me	thod Com	ponents				•
	R	eference Fr	ame Abso	lute				-
Supe	rsonic/Initial	Gauge Pres	sure [Pa]	0				•
		X-Ve	ocity [m/s	1.25				-
		Y-Ve	ocity [m/s] 0				•
	Turbulence	e						
	Speci	fication Met	hod <mark>K and</mark>	Epsilon				-
	Turbulen	t Kinetic En	ergy [m²/s	²] 0.001	.8			Ψ.
	Turbulent (Dissipation F	Rate [m²/s	³] 0.000	0963			•
			Apply	Close	Help			

Inlet Boundary Condition							
Variable	Variable $u(m/s)$ $v(m/s)$ $P(Pa)$ $k(m^2/s^2)$ $e(m^2/s^3)$						
Magnitude	1.25 0 - 0.0018 9.63e-05						
Zero Gradient Y							

6.7 Tree > Setup > Boundary Conditions > Zone > outlet. Change parameters as per below and click OK(Apply).

outlet								
Momentu	m Thermal	Radiation	Species	DPM	Multiphase	Potential	Structure	UDS
	Backflow F	Reference Fr	ame Abso	lute				-
		Gauge Pres	sure [Pa]	0				-
	Pressure	Profile Mult	iplier 1					-
Backflow	Direction Spe	cification Me	thod Norm	nal to Bo	undary			-
B	ackflow Press	ure Specific	ation Total	Pressu	e			-
Preve	ent Reverse Flo	w						
Avera	age Pressure S	pecification						
Targe	et Mass Flow F	Rate						
Tu	bulence							
	Spec	ification Met	hod Intens	sity and	Length Scale			•
	Backflow Tu	rbulent Inter	nsity [%] 3	.25				•
Ba	ckflow Turbule	ent Length S	cale [m] 0	.0035				-
			_					
			Apply	Close	Help			

Outlet Boundary Condition						
Variable u (m/s) v (m/s) P (Pa) Intensity (%) Length scale (m)						
Magnitude	Magnitude 0 3.25 0.0035					
Zero Gradient	Y	Y	-	-	-	

6.8 Make sure boundary condition type is wall for top and bottom walls.

Task Page 🛞	Task Page 🛞
Boundary Conditions	Boundary Conditions
Zone Filter Text E F F F F F F F F F F F F F F F F F F	Zone Filter Text 🔁 🗐 💭 bottom wall inter interior-surface_body outlet surface_body top_wall
Phase Type D mbture wall 6 Edit Copy Profiles Display Mesh Periodic Conditions	Phase Type D mixture V Vall V 7 Edit (Copy Profiles) Parameters Display Mesh Periodic Conditions

Wall Boundary Condition						
Variableu (m/s)v (m/s)P (Pa)k (m^2/s^2)e (m^2/s^3)						
Magnitude	0	0	-	0	0	
Zero Gradient	-	-	Y	-	-	

Outline View	<	Task Page	<
Filter Text		Reference Values	?
 Setup ֎ General ◆ ♥ Models 	^	Compute from	•
😑 🛃 Materials		Reference Values	_
😑 🗳 Fluid		Area [m ²] 0.25	
🚑 air		Density [kg/m ³] 1	
💽 💽 🚑 Solid		Depth [m] 1	
(*) 🖽 Cell Zone Conditions		Enthalov [1/kg] 0	=
Boundary Conditions		Length [m] 1	=
inlet			=
Inter (velocity-iniet, id=8) Internal		Pressure [Pa] 0	
Cutlet		Temperature [K] 288.16	
⊂ ≓ wall		Velocity [m/s] 1.25	
bottom_wall (wall, id=7)		Viscosity [kg/(m s)] 0.000147	
🧮 top_wall (wall, id=6)		Ratio of Specific Heats 1.4	
📁 Mesh Interfaces		Yolus for Heat Tran. Coef 200	-
🖉 Dynamic Mesh		Tpids for freat frail. Coel. (500	
Reference Values		Reference Zone	
🕑 🖾 Reference Frames			*
🕬 Named Expressions			

6.9 Tree > Setup > Reference Values. Change reference values as per below.

In case of '<u>Yplus for Heat Tran. Coef</u>' leave it as a default value (300)

6.10 **Tree > Solution > Methods.** Change the solution methods as per below.

Outline View	< Task Page
Filter Text	Solution Methods
 Setup General Models Materials Cell Zone Conditions Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Xeference Frames Named Expressions 	Pressure-Velocity Coupling Scheme SIMPLE Flux Type Rhie-Chow: distance based Auto Select Spatial Discretization Gradient Green-Gauss Cell Based Pressure Caused Order
 Solution Methods Controls Report Definitions Monitors Cell Registers Automatic Mesh Adaption Initialization Calculation Activities 	Second Order Momentum Second Order Upwind Turbulent Kinetic Energy Second Order Upwind Turbulent Dissipation Rate Second Order Upwind

6.11 Tree > Solution > Monitors > Residual. Change convergence criterions to 1e-05 and click OK(Apply).

Options	Equations				
✓ Print to Console	Residual	Monitor	Check Co	nvergence	Absolute Criteria
✓ Plot	continuity	✓		1	1e-05
Curves Axes	x-velocity			<u>/</u>	1e-05
Iterations to Plot	y-velocity		v	1	1e-05
1000 🌲	k	•		1	1e-05
Iterations to Store	epsilon	✓	•		1e-05
1000					
	Convergence Co	onditions.			
	Residual Values			Conver	gence Criterion
	Normalize	I	terations	absolu	te 💌
	✓ Scale		5		
	Compute Loc	al Scale			
	Renormalize				
	OK Plot	Cance	Help		

6.12 **Tree > Solution > Initialization**. Change parameters as per below and click **Initialize**.

Solution Initialization	?
Initialization Methods	
Hybrid Initialization Standard Initialization	
Compute from	
•	
Reference Frame	
Relative to Cell Zone Absolute	
Initial Values	
Gauge Pressure (pascal)	
0	
X Velocity (m/s)	
0.887	
Y Velocity (m/s)	
0	
Turbulent Kinetic Energy (m2/s2)	
0.0018	
Turbulent Dissipation Rate (m2/s3)	
9.63e-05	
Initialize Reset Patch	
Reset DPM Sources Reset Statistics	

6.13 Tree > Solution > Run Calculation. Change Number of Iterations to 10,000 and click Calculate.



6.14 Save your project and quit ANSYS fluent.

6.15 Duplicate the k-e setup for 10 degree half angle case to 4 degree angle case as per below then run the case. You need to make new connection between 4 degree case's mesh and duplicated setup. Once you enter the new setup, initialize first and then run.





6.16 Duplicate the k-e setup for 10 degree half angle and rename it as SST

6.17 Right click and select Edit....



6.18 **Tree > Setup > Models > Viscous**. Select SST model and use the default parameters as per below then click ok(Apply).

Model	Model Constants
Inviscid	Alpha*_inf
🔿 Laminar	1
 Spalart-Allmaras (1 eqn) 	Alpha_inf
🔿 k-epsilon (2 eqn)	0.52
🔿 k-omega (2 eqn)	Beta*_inf
 Transition k-kl-omega (3 eqn) 	0.09
Transition SST (4 eqn)	al
Reynolds Stress (5 eqn)	0.31
Scale-Adaptive Simulation (SAS) Data about Eddu Cimulation (SAS)	Beta i (Inner)
O Detached Eddy Simulation (DES)	0.075
Transition SST Options	Beta i (Outer)
Roughness Correlation	0.0828
Options	- ·
Curvature Correction	
Corner Flow Correction	User-Defined Transition Correlations
Production Kato-Launder	F_length
Production Limiter	hone
	none
	Re_thetat
	none
ОКСа	ncel Help

6.19 Tree > Solution > Controls. Change Under-Relaxation Factors as per below.

Solution Controls	?
Under-Relaxation Factors	
Density	^
Body Forces	
1	
Momentum	
0.5	
Turbulent Kinetic Energy	
0.5	
Specific Dissipation Rate	
0.5	
Intermittency	
0.5	
Momentum Thickness Re	
0.5	
Turbulent Viscosity	
1	
	-
Default	
Equations Limits Advance	d

6.20 Tree > Solution > Initialization > Initialize.

6.21 Tree > Solution > Run Calculation > Calculate.

After finish the calculation, **File > Save Project.** Then Close the window

6.22 Duplicate SST fluent setup for the 4 degree half angle case and run the simulation as per below (You should initialize before running the case).



7. Results (Read exercises (Section 8) before continuing.)

7.1 Creating lines for modified TKE and modified U plots.

Setting Up Domain > Surface > Create > Line/Rake. Create 7 lines at the given location on the table.

<u>F</u> ile	Domain		Physics User-D	efined Solution	Results Vie	w Pa	arallel Desigr		
		Mesh		Zones		Interfaces	Mesh Models	Adapt	Surface
Display	/		Scale	🚱 Combine 🝦 🕂 Delete	🕂 Append 🚽	Mesh	🛃 Dynamic Mesh	Refine / Coarsen	🕂 Create 🖕
(i) Info	- 🥨		🖉 🏒 Transform 🖕	🖵 Separate 🝦 📅 Deactivate	📇 Replace Mesh	Overset	💢 Mixing Planes		Zone
🖋 Units	. Check+	Quali	ity 👻 🖕 Make Polyhedra		Replace Zone		ổ Turbo Topology	👓 More 🚽	Partition
Outline Vie	w		Task Page	× .					Imprint
									Point
Filter Tex	t		Run Calculation	(?)					Line/Rake
 Setup 	moral		Check Case	Jpdate Dynamic Mesh					Plane. Line/Rake
⊕ © M	odels								Quadric
• 🖉 🖉 M	1aterials		Options						Iso-Surface
 ⊕ ⊞ E B B	ell Zone Cond oundary Cond	litions	Data Sampling for Ste	ady Statistics					Iso-Clip
💋 D 🔁 R	ynamic Mesh eference Value	es	Sampling Interval	ing Options					Transform

New Surface Na	me						
Options Line Reset	Type Line	•	Number of Points				
End Points		_					
x0 (m) 78		x1 (m) 78					
y0 (m) -3.52		y1 (m) 2					
z0 (m) 0		z1 (m) 0					
Select Points with Mouse							
Create Close Help							

Surface Name	x0	y0	x1	y1
Position-1	78	-3.52	78	2
Position-2	82	-4.23	82	2
Position-3	86	-4.9371	86	2
Position-4	98	-7.053	98	2
Position-5	102	-7.4	102	2
Position-6	110	-7.4	110	2
Position-7	118.5	-7.4	118.5	2

7.2 Defining custom field functions for modified U, modified TKE and skin friction coefficient.

User-Defined > **Custom**. Write the equation shown below and click **Define**. You will need to look up the Field function and the buttons to enter the parameters in the Definition. Definitions of the variables and custom field function that need to be defined are shown on table below.

<u>F</u> ile	Domain	Physics	User-De	fined	Solutio	on Resu	ilts
Field Func	tions	User I	Defined			Model Specific	
Custom		Eurotion H	ooko	💾 Mer	nory	😑 1D Coupling	
🖌 🖉 Units			JUKS	X Sca	lars	🕙 Fan Model	
Parame	Custom Create cust	tom field functions on	Demand	📑 Rea	d Table		

efinitior 0 * Vx + INV 0 5 (ew Fun	+ x - 60 - sin 1 6)	X Cos 2 7 PI me u*1	/ tan 3 8 e	y^x In 9	ABS log10 SQRT CE/C DEL		Select Operand Field Functions from Field Functions Mesh X-Coordinate Select	•
Define Manage Close Help								

Function Name	Definition
u*10+x (Modified U)	10*Vx+x-60
k*500+x (Modified TKE)	500*turb-kinetic-energy+x-60
skinfriction-coefficient	x-wall-shear * 2 / density / 1.25 ^ 2

7.3 Plotting modified U and modified TKE

Instruction for plotting modified U is given here. The only difference between modified U and modified TKE plot is a different "X-axis function".

Results > **Plots** > **XY Plot** > **Set Up...** > **Load File...** Select the 'Modified_u-10degree.xy' file downloaded from the class website and click **OK**

Note : Make su Turn off "Position on X	re about opti Axis" and T	ons as shown below. Furn on "Position on Y Axis"
Turn off "Position on X XY Plot Name xy-plot-1 Options Image: Position on X Axis Image: Position on Y Axis Image: Position on Y Axis Image: Write to File Image: Order Points File Data [1/1] Image: Profiles of modified velocity	Plot Direction X 0 Y 1 Z 0 Load File Free Data	Y Axis Function Direction Vector X Axis Function Custom Field Functions ▼ u*10+x Surfaces Filter Text = Filter Text ◆ Inlet ● Line-surface position-1 position-2 position-3
Save/Plot	: Axes Curves	position-4 position-5 position-6 position-7

You can compare EFD and CFD using the customizing functions (**Curves...**) on the lines you created as per below. Be careful about the axis location as shown below



7.4 Plotting skin friction coefficient

Results > **Plots** > **XY Plot** > **Load File...** Select the 'Skin_Friction_bot_wall.xy' file downloaded from the class website and click **OK**.

XY Plot Name		
xy-plot-2		
Options	Plot Direction	Y Axis Function
✓ Node Values	X 1	Custom Field Functions 💌
Position on X Axis	Y 0	skinfriction-coefficient 💌
Position on Y Axis	20	X Axis Function
		Direction Vector
File Data [1/1] File Data [1/1] File Data [1/1] File Data [1/1]	Load File Free Data	Surfaces Filter Text To For Filter Text To For Filter Text For Filter Text For Filter Text For Filter Text For Filter Fil
		New Surface
Save/Plot	Axes Curve	s) Close Help

Change the parameters as per below and click Plot.

You can change the axis by clicking **Axes...** under XY plot. Change the x-axis min and max to 60 and 130 respectively (uncheck Auto Range) and click **Apply**. Change the y-axis max and min to 4e-03 and -1e-03 respectively. Click **Apply** and click **Plot** again.



7.5 Total friction

Results > Reports > Forces. Select the zone where you want to calculate the total force then select print. This will print a report as per below

Options Forces Moments Center of Pressure Save Output Parameters	e Direction Vector	Wall Zones Filter Text	
	Print	te) Close Help	

Forces Zone bottom_wall	Forces (n) Pressure (1.058987 38.91	7494 0)		Viscous (0.34016388 -0.00	010073304 0)		Total (1.3991509 38.916487 0)
Net	(1.058987 38.917494 0)			(0.34016388 -0.0010073304 0)			(1.3991509 38.916487 0)
Forces - Direction Vector Zone	(1 0 0) Forces (n) Pressure	Viscous	Total	Coefficients Pressure	Viscous	Total	
Net	1.058987	0.34016388	1.3991509	5.4220134 5.4220134	1.7416391	7.1636525	

7.6 Finding the pressure difference between inlet and outlet.

You can simply write pressure at bottom wall to a file and take the difference of pressure at inlet and outlet.

xy-plot-3			
Options	Plot Direction	Y Axis Function	
Vode Values Vosition on X Axis Position on Y Axis Vosition on Y Axis V	X 1 Y 0 Z 0	Pressure Static Pressure X Axis Function Direction Vector Surfaces Filter Text Inlet Inlet Uine-surface Outlet Wall bottom_wall top_wall New Surface) =



7.7 Plotting contours, velocity vectors and streamlines.

Scale Skip

0.03

Vector Options...

Custom Vectors...

Colormap Options..

Fluid
 Inlet
 Internal
 Line-surface

New Surface

Save/Display Compute Close Help

Une sc
 Outlet
 Wall

\$ 0

Refer to previous manuals for lab 1 and 2 for plotting streams, velocity vectors and pressure distributions. You can change the scales and levels for vectors and streamlines respectively to show the separation region. Few examples are shown at below.



6.53e-01

5.81e-01 5.08e-01 4.36e-01

3.64e-01

2.92e-01 2.20e-01

1.48e-01

7.62e-02 4.20e-03 m/s]

5 (m)



8. Data Analysis and Discussion

8.1 Simulation of turbulent diffuser flows without separation (4 degree) (+20)

- 8.1.1 Run simulations for 4 degree half angle diffuser with k- ε model.
- 8.1.2 Run simulations for 4 degree half angle diffuser with SST model.
- 8.1.3 Questions:
- Do you observe separations in 8.1.1 or 8.1.2? (use streamlines)
- What are the differences between 8.1.1 and 8.1.2 regarding modified u, modified TKE, and the variables in the following table?

Turbulent model	Total pressure difference between inlet and outlet (Pa)	Total friction force on the upper wall (N)
SST		
k-e		
Relative error (%)		

- Figures need to be reported (for both 8.1.1 and 8.1.2):
 (1) Residual history (2) Modified u vs. x (3) Modified TKE vs. x (4) Contour of pressure
 (5) Contour of axial velocity (6) Velocity vectors and streamlines
- **Data need to be reported**: the above table with values.

8.2 Simulation of turbulent diffuser flows with separation (10 degree) (+22):

- 8.2.1 Run simulations for 10 degree half angle diffuser with k- ε model.
- 8.2.2 Run simulations for 10 degree half angle diffuser with SST model.
- 8.2.3 Questions:
- Do you observe separations in 8.2.1 or 8.2.2? (using streamlines)
- Comparing with EFD data, what are the differences between 8.2.1 and 8.2.2 on the following aspects: (1) Modified velocity (2) Modified TKE (3) Skin friction factor on top and bottom walls (4) Variables in the following table.

Turbulent models	Total pressure difference between inlet and outlet (Pa)	Total friction force on the upper wall (N)	
SST			
k-e			
Relative error (%)			

- If any separation shown, where is the separation point on the diffuser bottom wall (x=?) and where does the flow reattach to the diffuser bottom wall again (x=?) (use wall friction factor)
- Do you find any separation on the top wall?
- **Figures need to be reported** (for both 8.2.1 and 8.2.2):

(1) Residual history (2) Modified u vs. x with EFD data (3) Modified TKE vs. x with EFD data (4) Skin friction factor distributions on top and bottom walls with EFD data (5) Contour of pressure (6) Contour of axial velocity (7) Velocity vectors and streamlines with appropriate scales showing the separation region if the simulation shows separated flows.

• **Data need to be reported:** The above table with values.

8.3 Questions need to be answered in CFD Lab3 report

- 8.3.1 Questions in exercises 8.1-8.2.
- 8.3.2 By analyzing the results from exercise 1 and exercise 2, what can be concluded about the capability of k- ε and SST models to simulate turbulent flows inside a diffuser with and without separations? (+3)

9. Grading scheme for CFD Lab Report

(Applied to all CFD Lab reports)

Section

Section			Points
1	Title Page		5
	1.1 Course Name		
	1.2 Title of report		
	1.3 Submitted to "Instructor's name"		
	1.4 Your name (with email address)		
	1.5 Your affiliation (group, section, department)		
	1.6 Date and time lab conducted		
2	Test and Simulation Design		10
	Purpose of CFD simulation		
3	CFD Process		20
	Describe in your own words how you implemented CFD process		
	(Hint: CFD process block diagram)		
4	Data Analysis and Discussion Exection 8 (Page# 47) for CFD Lab 3		45
	Answer questions given in Exercises of the CFD lab handouts		
5	Conclusions		20
	Conclusions regarding achieving purpose of simulation		
	Describe what you learned from CFD		
	Describe the "hands-on" part		
	Describe future work and any improvements		
		Total	100

Additional Instructions:

- 1. Each student is required to hand in individual lab report.
- 2. Conventions for graphical presentation (CFD):
 - * Color print of figures recommended but not required
- 3. Reports will not be graded unless section 1 is included and complete