# Simulation of Turbulent Flow in an Asymmetric Diffuse

#### 58:160 Intermediate Mechanics of Fluids CFD LAB 3

By Timur K. Dogan, Michael Conger, Maysam Mousaviraad, 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 using CFD Educational Interface (ANSYS). 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.

# 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 the sketch window you will see in FlowLab with definitions for all geometry parameters. Before the diffuser, a straight channel was used for generating fully developed channel flow at the diffuser inlet. The origin of the coordinates is placed at the inlet of the channel before diffuser.



In CFD Lab3, all EFD data for turbulent airfoil flow in this Lab will be provided by the TA and saved on the Fluids lab computers.

# 3. Open ANSYS Workbench Template

- 3.1. Download CFD Lab 3 Template from class website.
- 3.2. Open Workbench Project Zip file simply by double clicking file. This file contains all the systems that must be solved for CFD Lab 3.



# 4. Geometry Creation

4.1 Right click Geometry and select New Geometry. (The geometries for both cases are linked together therefore only one geometry needs to be generated.)



4.2 Select Meter and click OK.

ANSYS Workbench		x
Select desired length unit:		
Meter	Foot	
O Centimeter	🔘 Inch	
O Millimeter		
O Micrometer		
Always use project ur	nit	
Always use selected	unit	
Enable large model su	oport	
ОК		

4.3 Select XYplane and click New Sketch button.

政 A: Fluid Flow (FLUENT) - DesignModeler
File Create Concept Tools View Help
] 🔄 📑 📑 📫 ] 💬 Undo 📿 Redo 🛛 Select: 🍡
■ • h • /1 • h • /3 • /x •
🚽 XYPlane 🔹 🗚 🛛 Sketch1 🔹 💋 🗍 🧚 Gen
Tree Outline New Sketch
⊢, 🚱 A: Fluid Flow (FLUENT)
i
Sketch1
ZXPlane
VZPlane
📖 🖓 0 Parts, 0 Bodies

4.4 Right click Sketch1 and select Look at.

Tree Outline	ų.
🖃 🗤 🖓 A: Fluid Flow (FLU	JENT)
🚊 🗸 🖈 XYPlane	_
Sketch1	l
	Always Show Sketch
YZPlane	Hide Sketch
🦾 🖓 🕼 🖉 🖉 🖉 🖉 Ints, 0 Bo	👰 Look at
	🚉 Show Dependencies
	X Delete
	🧚 Generate
	a]b Rename

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

	Draw		
Line		l	
< Tangent Line			
Line by 2 Tangen	te		
A Polyline			
Debreen			
- Polygon			
Rectangle by 3 D	ointe		
A Rectangle by 5 PC	oints		
Circle			
Circle by 3 Tange	ante		
Arc by Tangent			
Arc by 3 Points			
Arc by Center			
Fillinse			
2 Spline			
* Construction Poi	int		
& Construction Poi	nt at Intersecti	on	
	Modify		
	Dimension	IS	
	Constraint	s	
	Settings		
Sketching Modeling	g		



4.6 Sketching > Dimensions > General. Click on the vertical line then click on the left side of the line. Change the dimension to 2m.



4.7 Sketching > Draw > Line. Create a horizontal line on the x-axis as per below (H indicates that line is horizontal).



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



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



4.10 Sketching > Dimensions > Angle. Select the line circled in red below then select the x-axis then change the angle to 10 degrees.



De	etails View	<del>4</del>
	Details of Sketch1	
	Sketch	Sketch1
	Sketch Visibility	Show Sketch
	Show Constraints?	No
	Dimensions: 3	
	A3	10 °
	H2	60 m
	V1	2 m
Ξ	Edges: 3	
	Line	Ln7
	Line	Ln8
	Line	Ln9

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



4.12 Sketching > Dimensions > General. Change the length of the line circled in red to 70m.



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



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



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



4.16 Sketching > Constraints > Equal Distance. Click on point number 1 and click on the point which lies on the same line that intersects with the y-axis then click on point number 2 and the y-axis.



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



4.18 Sketching > Constraints > Equal Length. Click on the two lines circled in red as shown below.



4.19 Sketching > Draw > Line. Draw the final line circled in red as shown below.



4.20 Sketching > Dimensions > General. Change the length of the line circled in red to 9.4m.



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

4.21 Concept > Surface From Sketches. Select the sketch you created and click apply then click generate button. This will create a surface as shown below.

📖 A: Fluid Flov	w (FLUENT) - DesignModeler	💓 A: Fluid Flow (FLUENT) - DesignModeler
File Create	Concept Tools View Help	File Create Concept Tools View Help
J VZPlane Tree Outline □ √ ∞ A: Flu □ √ ∞ A: Flu	<ul> <li>Lines From Points</li> <li>Lines From Sketches</li> <li>Lines From Edges</li> <li>3D Curve</li> <li>Split Edges</li> <li>Surfaces From Edges</li> <li>Surfaces From Sketches</li> <li>Surfaces From Faces</li> </ul>	Image: Select:     Image: Select
<u></u> *	Cross Section	
	Details View	<b>4</b>
	Details of SurfaceSk2	
	Surface From Sketches	SurfaceSk2





4.22 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.

File Create Concept       Tools       View Help         Image: Concept       Treeze       Image: Concept       Image: Concept         Image: Concept       Image: Concept       Image: Concept       Image: Concept       Image: Concept         Image: Concept       Image: Concept       Image: Concept       Image: Concept       Image: Concept       Image: Concept         Image: Concept       Image: Concept       Image: Concept       Image: Concept       Image: Concept       Image: Concept<	
Image: Constraint of the second se	
III • $h$ • $h$ • $f$ Image: Constraint of the second se	
XYPlane <ul> <li>Mamed Selection</li> <li>Attribute</li> <li>Attrite</li> <li>Attribute</li> <li>Attribute</li></ul>	
Tree Outline	
😑 – 🖉 🐼 A: Fluid Flow (F 🧠 Mid-Surface	
└───★ XYPlane ▲ Joint	
Sketc 🕘 Enclosure	
Image Split	
Symmetry	
- v 1 Part, 18	
Surfa Cui Surface Film	
Solid Evension (Beta)	
B Connect	
Projection Details View	<b></b> д
Repair	_
Analysis Tools   Face Split FaceSplit1	
Form New Part Face Split Group 1 (RMB)	
Face Split Type By Points and Edges	
Electronics Target Face Apply Cancel	
Addins Tool Geometry 0	_
Ø Options	
· · ·	

4.23 Click on the yellow region shown below.

De	etails View	<b>#</b>
Ξ	Details of Faces	Split1
	Face Split	FaceSplit1
	Face Split Grou	p1 (RMB)
	Face Split Type	By Points and Edges
	Target Face	1
	Tool Geometry	0

4.24 While holding control button click on the two points circled in red then click apply button.

Sketching Mod	eling	
etails View		ť
Details of FaceS	Split1	
Face Split	FaceSplit1	
Face Split Grou	p1 (RMB)	
Face Split Type	By Points and Edges	
Target Face	1	
Tool Geometry	Apply	Cancel
	Sketching Mod etails View Details of Faces Face Split Face Split Grou Face Split Type Target Face Tool Geometry	Sketching Modeling etails View Details of FaceSplit1 Face Split FaceSplit1 Face Split Group 1 (RMB) Face Split Type By Points and Edges Target Face 1 Tool Geometry Apply

4.25 Click on the region marked with red rectangle below.

De	etails View	7
Ξ	Details of Faces	Split1
	Face Split	FaceSplit1
Ξ	Face Split Grou	p1 (RMB)
	Face Split Type	By Points and Edges
	Target Face	1
	Tool Geometry	1

4.26 While holding control button click on the two points circled in red then click apply button.

Sketching Modeling	
Details View 4	
Details of FaceSplit1	
Face Split FaceSplit1	
Face Split Group 1 (RMB)	
Face Split Type By Points and Edges	
Target Face 1	
Tool Geometry Apply Cancel	

4.27 Click the generate button then close the window and update geometry.



# 5. Mesh Generation

5.1 Right click on Mesh and click Edit. (The meshes for both cases are linked together therefore only one mesh needs to be generated.)



5.2 Right click on Mesh then select Insert then Mapped Face Meshing.



5.3 Select all three surface as per below and click apply.

						÷ +	
							Y
1							
1	Details of "Mapped Face Meshing	" - Mapped Face Meshing 🛛 🕂					<b>↑</b>
	Scope						•
	Scoping Method	Geometry Selection					
	Definition	Apply Cancel					
ľ	Suppressed	No		0.00	30.00	60.00 (m)	
	Method	Ouadrilaterals			15.00	45.00	
	Radial Number of Divisions	Default			15.00	40.00	
	Constrain Boundary	No	Geometry / Print Preview /				

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



5.5 Right click on mesh and Insert then Sizing.

Geometry	4:35 PN
insert @ Method Update & Sizing Generate Mesh	
Preview Show Show Create Pinch Controls Clear Generated Data	

5.6 While holding control click on the edge shown below and click apply.

$\wedge$	$\land$	
	( <b>-</b> )	
U U	$\nabla$	

Details of "Sizing" - Sizing 4					
-	Scope				
	Scoping Method	Geometry Selection			
	Geometry	Apply	Cancel		
-	Definition				
	Suppressed	No			
	Туре	Element Size			
	Element Size	Default			
	Behavior	Soft			
	Curvature Normal Angle	Default			
	Growth Rate	Default			

5.7 Change parameter for edge sizing as per below and click apply.

D	Details of "Edge Sizing" - Sizing 4					
	- Scope					
	Scoping Method	Geometry Selection				
	Geometry	Apply	Cancel			
Ξ	Definition					
	Suppressed	No				
	Туре	Number of Divisions				
	Number of Divisions	59				
	Behavior	Hard				
	Bias Type					
	Bias Factor	15.106				
			Ť•			

- 5.8 Right click on mesh and Insert then Sizing.
- 5.9 While holding control click on the edge shown below and click apply.



5.10 Change parameter for edge sizing as per below and click apply.

De	Details of "Edge Sizing 2" - Sizing 🛛 🛛 🕂					
Details of "Edge Sizing 2" - Sizing       Scope       Scoping Method     Geometry Selection       Geometry     Apply       Cancel       Definition       Suppressed     No       Type     Number of Divisions				1		
	Scoping Method	Geometry Selection	g metry Selection Apply Cancel  ber of Divisions d			
	Geometry	Apply	Cancel	]		
-	Definition					
	Suppressed	No				
	Туре	Number of Divisions				
	Number of Divisions	59				
	Behavior	Hard				
	Bias Type					
	Bias Factor	87.76		1		



- 5.11 Right click on mesh and Insert then Sizing.
- 5.12 While holding control 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	Apply	Cancel		
	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	59			
	Behavior	Hard			
	Bias Type				
	Bias Factor	3.6776			

- 5.14 Right click on mesh and Insert then Sizing.
- 5.15 While holding control click on the edge shown below and click apply.



5.16 Change parameter for edge sizing as per below and click apply.

Scope					
	Scoping Method	Geometry Selection			
	Geometry	Apply	Cancel		
-	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	59			
	Behavior	Hard			
	Bias Type				
	Bias Factor	1.8593			



- 5.17 Right click on mesh and Insert then Sizing.
- 5.18 While holding control click on the edge shown below and click apply.



5.19 Change parameter for edge sizing as per below and click apply.

Details of "Edge Sizing 5" - Sizing #					
Ξ	Scope				
	Scoping Method	Geometry Selection			
	Geometry	Apply	Cancel		
	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	59			
	Behavior	Hard			
	Bias Type				
	Bias Factor	4.3763			



5.20 Click the Generate Mesh button.

A : Fluid Flow (FL	UENT) - Meshing [ANS)	/S Academic Teaching	Introductory]
File Edit View	Units Tools Help	誟 Generate Mesh	ta 💀 🖬 📬 🕶
戸 Show Vertices	Wireframe E	dge Coloring 👻 🄏 🕻	1- 1- 1-
Mesh 😏 Update	🔞 Mesh 🔻 🔍 Mesl	h Control 👻 🔄 🗍 Met	ric Graph   🖏 Opti

5.21 Select Geometry and click edge button.





5.22 While holding the control button select the top edges and right click on it then select Create Named Selection. Change the name to top\_wall and click OK. Similarly name the bottom\_wall, inlet and outlet.

		Go To
		Parts
		Set
	0.00 30.00 60.00 (m)	Suppress Body           Image: Hide Body
	15.00 45.00	Create Coordinate System
ometry / Print Preview /		Refresh Geometry
	Selection Name  Enter a name for the selection group: top_wall  Apply selected geometry Apply geometry items of same: Size Type Location X Location Y Location Z  OK Cancel	

5.23 Close window and update the mesh

# 6. Setup

6.1 Right click setup and click edit.



6.2 Check Double Precision and select OK.

E FLUENT Launcher (Setting Edit Only)			
ANSYS	FLUENT Launcher		
Dimension ② 2D ③ 3D	Options Double Precision Use Job Scheduler		
Display Options          Image: Display Mesh After Reading         Image: Display Mesh After R	Processing Options		
	ancel Help -		

6.3 Problem Setup > General. Set the parameters as per below.

Problem Setup	General			
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Mesh Scale Check Report Quality Display Solver			
Dynamic Mesh Reference Values Solution	Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative		
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities	Time ◉ Steady ⓒ Transient	2D Space Planar Axisymmetric Axisymmetric Swirl		
Run Calculation Results	Gravity	Units		
Graphics and Animations Plots Reports	Help			

6.4 Problem Setup > Models > Viscous > Edit. Select parameters as per below and click OK. (You will need to solve with SST model with default settings as well.)

Problem Setup	Models	Viscous Model	
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Models Multiphase - Off Energy - Off Viscous - Laminar Radiation - Off Heat Exchanger - Off Species - Off Solidification & Melting - Off Acoustics - Off Edit Help	Model         Inviscid         Laminar         Spalart-Allmaras (1 eqn)         & k-epsilon (2 eqn)         Transition k-kl-omega (3 eqn)         Transition SST (4 eqn)         Reynolds Stress (5 eqn)         Scale-Adaptive Simulation (SAS)         k-epsilon Model         @ Standard         Realizable         Near-Wall Treatment         Standard Wall Functions         @ Enhanced Wall Functions         @ Enhanced Wall Treatment         User-Defined Wall Functions         Enhanced Wall Treatment Options         If Pressure Gradient Effects	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 User-Defined Functions Turbulent Viscosity none Prandtl Numbers TKE Prandtl Number TDR Prandtl Number none  Cance Help Cance Help

6.5 Problem Setup > Materials > Fluid > air > Create/Edit. Change the fluid properties and then click Change/Create then close window

Problem Setup	Materials	Create/Edit Mate	erials			<b>x</b>
General Models Materials	Materials Fluid	Name		Material Type fluid	•	Order Materials by
Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Beference Values	Solid aluminum	Chemical Formula		FLUENT Fluid Materials air Mixture none	•	
Solution Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations		Properties Density (kg/m3) ( Viscosity (kg/m-s) (	constant 1 constant 0.000147	Edt		_
Plots Reports	Create/Edit Delete		Change/Create	Delete Clos	e Help	

6.6 Problem Setup > Boundary Conditions > Zone > inlet > Edit. Change parameters as per below and click OK.

Problem Setup	Boundary Conditions	Velocity Inlet
General Models Materials	Zone bottom_wall	Zone Name inlet
Phases Cell Zone Conditions Boundary Conditions	interior-surface_body outlet surface_body	Momentum Thermal Radiation Species DPM Multiphase UDS
Mesh Interfaces Dynamic Mesh Reference Values	top_wall	Veloaty Specification Method Components
Solution		Supersonic/Initial Gauge Pressure (pascal) 0 constant
Solution Controls Monitors		X-Velocity (m/s) 1.25 constant V
Solution Initialization Calculation Activities Run Calculation		Y-velocity (m/s) 0 constant
Results	Phace Type TD	Specification Method K and Epsilon
Plots Reports	mixture velocity-inlet 10015	Turbulent Kinetic Energy (m2/s2) 0.0018 constant
	Edit Copy Profiles	Turbulent Dissipation Rate (m2/s3) 9,63e-05
	Display Mesh Periodic Conditions	OK Cancel Help

6.7 Problem Setup > Boundary Conditions > Zone > outlet > Edit. Change parameters as per below and click OK.

Problem Setup	Boundary Conditions	Pressure Outlet
General Models Materials Phases Cell Zone Conditions Boundary Conditions Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Zone Dottom_wall interior-surface_body outlet surface_body top_wall Phase mixture Type ID ID ID ID ID ID ID ID ID ID	Zone Name outiet Momentum Thermal Radiation Species DPM Multiphase UDS Gauge Pressure (pascal) 0 constant • Backflow Direction Specification Method Normal to Boundary • Average Pressure Specification Target Mass Flow Rate Turbulence Specification Method K and Epsilon • Backflow Turbulent Kinetic Energy (m2/s2 Backflow Turbulent Dissipation Rate (m2/s2 0.0035 constant •
	Display Mesh Periodic Conditions	

6.8 Problem Setup > Reference Values. Change reference values as per below.

Problem Setup	Reference Values	
General Models Materials Phases	Compute from	•
Cell Zone Conditions Boundary Conditions	Area (m2)	0.25
Mesh Interfaces Dynamic Mesh	Density (kg/m3)	1
Reference Values Solution	Depth (m)	1
Solution Methods Solution Controls	Enthalpy (j/kg)	0
Monitors Solution Initialization	Length (m)	1
Run Calculation	Pressure (pascal)	0
Graphics and Animations	Temperature (k)	288.16
Plots Reports	Velocity (m/s)	1.25
	Viscosity (kg/m-s)	0.000147
	Ratio of Specific Heats	1.4
	Reference Zone	

## 7. Solve

7.1 Solution > Solution Methods. Change the solution methods as per below.

Problem Setup	Solution Methods	
General Models Materials Phases Cell Zone Conditions	Pressure-Velocity Coupling Scheme SIMPLE   Sobatial Discretization	
Mesh Interfaces Dynamic Mesh Reference Values	Gradient Green-Gauss Cell Based	Â
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Pressure Second Order ▼ Momentum Second Order Upwind ▼ Turbulent Kinetic Energy Second Order Upwind ▼ Turbulent Dissipation Rate	Ш
Results Graphics and Animations Plots Reports	Second Order Upwind       Transient Formulation	Ŧ

7.2 Solution > Monitors > Residuals > Edit. Change convergence criterions to 1e-5 and click OK.

Problem Setup	Monitors	Residual Monitors		×
General	Residuals, Statistic and Force Monitors	Options	Equations	
Models	Residuals - Print, Plot	Print to Console		1e-3 A
Materials	Statistic - Off	I Plot	x-velocity	1e-5
Cell Zone Conditions	Drag - Off	Window		
Boundary Conditions	Moment - Off		y-velocity 🔽	1e-5
Mesh Interfaces		Curves Axes		
Dynamic Mesh	Edit	Iterations to Plot	K V	1e-5
Reference Values		1000	epsilon	1e-5
Solution	Surface Monitors			· · ·
Solution Methods			Residual Values	Convergence Criterion
Solution Controls		Iterations to Store	Normalize Iterations	absolute 👻
Monitors Solution Initialization		1000	5	
Calculation Activities				
Run Calculation			Scale	
Results	Create Edit Delete		Compute Local Scale	
Graphics and Animations	Volume Monitors			
Plots		OK Pla	ot Renormalize Cancel	Help
Reports				

7.3 Solutions > Solution Initialization. Change parameters as per below and click initialize.

Problem Setup	Solution Initialization
General Models Materials Phases Cell Zone Conditions Boundary Conditions	Initialization Methods           Hybrid Initialization           Standard Initialization           Compute from
Mesh Interfaces Dynamic Mesh Reference Values	Reference Frame     @ Relative to Cell Zone
Solution Solution Methods	Absolute  Initial Values
Solution Controls Monitors Solution Initialization Calculation Activities Pun Calculation	Gauge Pressure (pascal)
Results	x velocity (m/s) 0.887
Plots Reports	Y Velocity (m/s)
	Turbulent Kinetic Energy (m2/s2) 0.0018
	Turbulent Dissipation Rate (m2/s3) 9.63e-05
	Initialize Reset Patch Reset DPM Sources Reset Statistics

7.4 Solution > Run Calculation. Change number iterations to 10,000 and click Calculate.

Problem Setup	Run Calculation	
General Models	Check Case	Preview Mesh Motion
Materials Phases	Number of Iterations	Reporting Interval
Cell Zone Conditions	10000	1
Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Profile Update Interval	
Solution	Data File Quantities	Acoustic Signals
Solution Methods Solution Controls Monitors Solution Initialization	Calculate	]
Calculation Activities Run Calculation	Help	
Results		
Graphics and Animations Plots Reports		



# 8. Post Processing

8.1 Surface > Line/Rake. Create 7 Lines at the given location on the table.

Surface	Display R	eport Parallel	View	Help			
Zor	ne	💶 Line	/Rake S	urface			8
Par	tition	Options	;	Туре	Numbe	er of Points	
Poi	nt	🗆 Lin	e Tool	Line 🔹	10		
Lin	e/Rake	Re	eset				
Pla	ne	End Poi	nts				
Qu	adric	x0 (m			v1 (m)		
Iso	-Surface	~~ ("	/ /8		~* (***)	78	
Iso	-Clip	y0 (m	-3.52		y1 (m)	2	
Tra	nsform	20 (m	0		z1 (m)	0	
Ma	nage			Select Point	s with Mo	use	
		New Su	rface Nar	me			
		positio	on-1				
			reate	Manage	Clos	se He	slp

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

8.2 Define > Custom Field Function. Write the equation shown below and click Define. You will need to use the Field function and the buttons to enter the parameters. Definitions of the variables and custom field function that need to be defined are shown on table below.

File Mesh	Define	Solve	Adapt	Surface	Display
Problem Setur General Models Materials Phases Cell Zone Cc Boundary C Mesh Interf Dynamic Me Reference V Solution Cor Monitors Solution Cor Monitors Solution Init Calculation , Run Calcula Besults	Gei Ma Ph. Cei Bo Op Me Dyn Me Mit Tu Inju DT	neral odels ases Il Zone ( undary ( erating ish Inter namic N esh Mor king Pla rbo Top ections RM Ray	Condition Condition Condition faces faces faces faces ology s	ns ns ins	
Graphics an Plots	She	ell Cond	luction V	Valls	
Reports	Pai Pro Un Use	ameter: ofiles its er-Defin	ed		ŀ

Custom Field Function Calculator	144.0	×	
Definition           10 * Vx + x - 60           +         -         X         / y^x         ABS           INV         sin         cos         tan         In         log10           0         1         2         3         4         SQRT           5         6         7         8         9         CE/C           (         )         PI         e         .         DEL           New Function Name         u*10+x         .         .         .	Select Operand Field Functions from         Field Functions         Mesh         X-Coordinate         Select		
Define Manage Close Help			

Function Name	Definition	
u*10+x	10*Vx+x-60	
k*500+x	500*turb-kinetic-energy+x-60	

8.3 Refer to previous instruction for plotting streams, velocity vectors and pressure distributions. Additionally you can compare EFD and CFD using the custom field functions as per below.



# 9. Exercises

You need to complete the following assignments and present results in your lab reports following the lab report instructions.

# **Simulation of Turbulent Flow in an Asymmetric Diffuser**

- You can save each case file for each exercise using "file" → "save as"
- Otherwise stated, use the parameters shown in the instruction.

#### 1. Simulation of turbulent diffuser flows without separation

- a. Iterate until the solution converges using the default values in the instructions, EXCEPT for the following parameters:
  - Diffuser half angle: 4 degree
  - Viscous model: v2f.
- b. Repeat 1.1 but use the k-ε model.
- c. Questions:
  - Do you observe separations in 1.1 or 1.2? (use streamlines)
  - What are the differences between 1.1 and 1.2 regarding modified u, modified TKE, and the variables in the following table?

Turbulent model	Total pressure difference (Pa)	Total friction force on the upper wall (N)
V2f		
k-e		
<b>Relative error (%)</b>		

- **Figures to be saved** (for both 1.1 and 1.2): 1. XY plots for residual history, modified u vs. x and modified TKE vs. x, 2. Contours of pressure and contours of axial velocity.
- **Data to be saved:** the above table with values.

## 2. Simulation of turbulent diffuser flows with separation:

- d. Iterate until the solution converged using the default values in the instructions, except the following parameters:
  - Diffuser half angle: 10 degrees
  - Viscous model: v2f
- e. Repeat 2.1 but use the k-ε model.
- f. Questions:
  - Do you observe separations in d or e? (using streamlines)
  - Comparing with EFD data, what are the differences between 2.1 and 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	Total friction	
	difference	force on the	

	upper wall
V2f	
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=?) (hint: use wall friction factor)
- Do you find any separation on the top wall?
- **Figures to be saved** (for both 2.1 and 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 to be saved: The above table with values.

### 3. Questions need to be answered in CFD Lab3 report:

- g. Questions in exercises 1-2.
- h. By analyzing the results from exercise 1 and exercise 2, what can be concluded about the capability of k- $\varepsilon$  and v2f models to simulate turbulent flows inside a diffuser with and without separations?