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

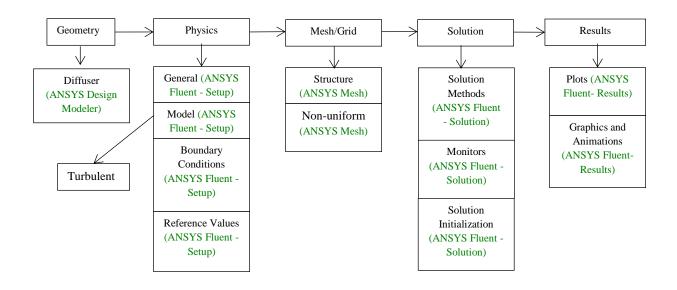
#### ME:5160 Intermediate Mechanics of Fluids CFD LAB 3 (ANSYS 2020 R2; Last Updated: August. 23, 2020)

By Timur Dogan, Michael Conger, Dong-Hwan Kim, Sung-Tek Park, 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.



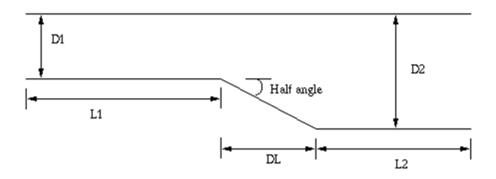
Flow Chart for "CFD Process" for diffuser

### 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- $\epsilon$ .

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

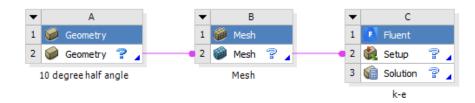
Table 1 – Main particulars



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.

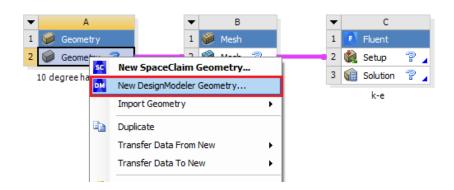


3.2 File > Save. Save the project on the network drive and Call it "*CFD Lab 3*".

# 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.

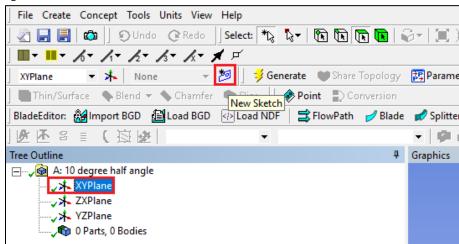
4.1 Right click Geometry and select New DesignModeler Geometry...



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

File Create Concept Tools	Units View Help	
] 🖉 🔒 📕   🖚  ] 💬 Unda	✓ Meter	te te te te 💽 🖉
<b>■</b> ▼ <b>■</b> ▼ <b>/</b> 1▼ <b>/</b> 2▼	Centimeter	
XYPlane 🔻 📥 None	Millimeter Micrometer	🖤 Share Topology 🛛 😰 P
📕 🖿 Thin/Surface – 💊 Blend 🔻		nt 🚽 Conversion
BladeEditor: 🔏 Import BGD	Inch	FlowPath 🥑 Blade 💋
] ❷ 巫 S ≣ ( 寮 ⊉	Large Model Support	•
Tree Outline		4 Grap
🖃 🗸 🚱 A: 10 degree half angle	✓ Degree	
XYPlane	Radian	
·····↓ XXPIane	Model Tolerance	
👘 0 Parts, 0 Bodies		

4.3 Select XYplane and click New Sketch button.



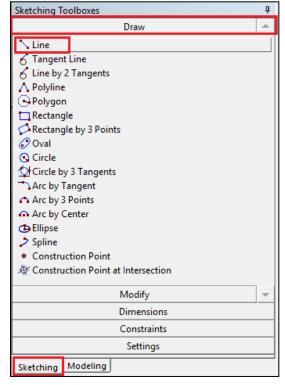
#### 4.4 Right click Sketch1 and select Look at.

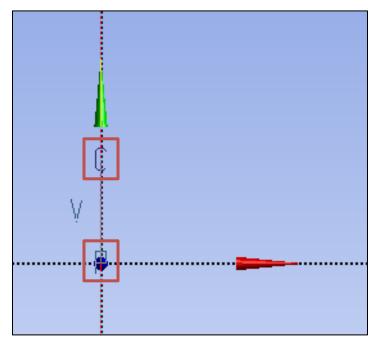
Tree Outline	ф.
🖃 🗤 🏑 🎯 A: 10 degree half angle	
🚊 🗸 XYPlane	
Sketch1	
ZXPlane V Always Show Sketch	
YZPlane P Hide Sketch	
👘 0 Parts, 0 B 🕂 Look at	
Show Dependencies	
X Delete	
🥩 Generate (F5)	
allo Rename (F2)	

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

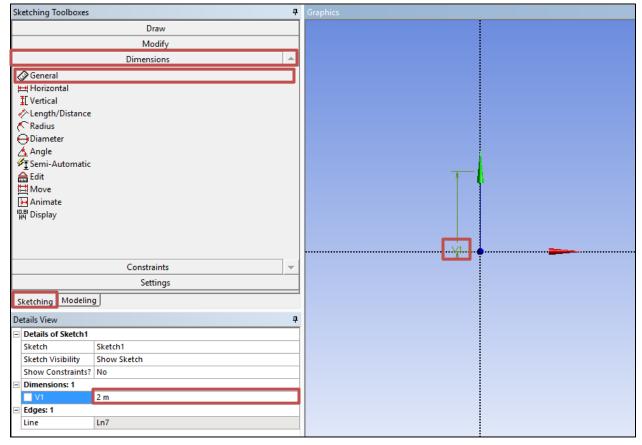
File	Create Con	cept Tools	Units View	Help			
	🔒 🛃 🗠	S Undo	Redo	Select: *[	§ 🔤 🖹		
-	- 1- 7-	1- 12-	/s= /x= ;				
XYP	lane 🔻	📥 Sket	ch1 🔻	翘 🛛 誟	Generate 🐚	Share Topo	logy
	Thin/Surface	Nend 👻	🔦 Chamfer	Slice 🖤	🔷 🔗 Point	Convers	ion
Blade	Editor: 🔏 Im	port BGD	🖹 Load BGD	<i>⊲&gt;Load N</i>	DF 🛛 式 Flo	wPath 🥑 I	Blade
\$	<b>⊼</b> 3 ≣	(函捷		•			
Sketcl	hing Toolboxes						ą.
			Draw	,			
			Modi	fy			
			Dimensi	ons			
			Constraint	ts			
777 Fi	xed						
<del>,,,,</del> H	orizontal						
	ertical						
	erpendicular						
-	angent						
-	oincident						
-	lidpoint						
	mmetry						
-	arallel						
_	oncentric						
	qual Radius						
	qual Length						
	qual Distance						=
CON A	uto Constraint	s			Globa	l: 🗌 Cursor:	<b>V</b>
							_
			Settings				-
Sketo	hing Modelin	ng					
	s View						ą
- Det	ails of Sketch1						
Ske	tch	Sketch1					
Ske	tch Visibility	Show Sket	th				
	w 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).

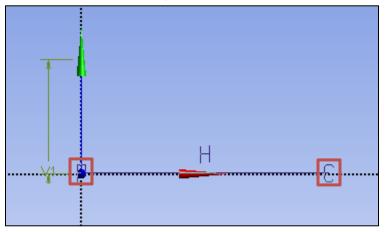




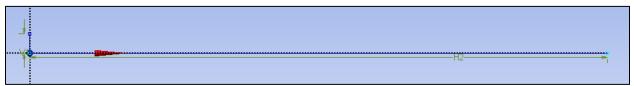
4.7 Sketching > Dimensions > General. Click on the vertical line then click on the left side of the line to place the dimension. Change the dimension in Details View to 2m (skip the unit ([m]) when put in the value).



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).



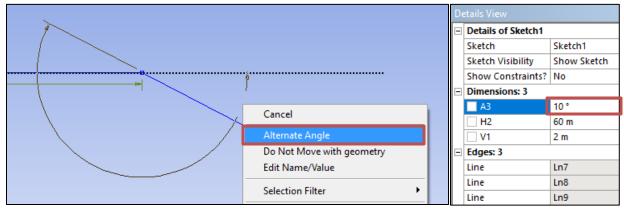
4.9 **Sketching** > **Dimensions** > **General**. Change the length of the horizontal line you created to 60m.



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



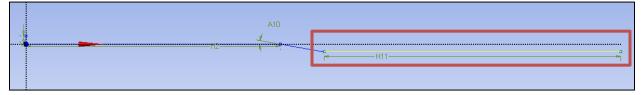
4.11 **Sketching** > **Dimensions** > **Angle**. Select the line just created then select the x-axis then change the angle to 10°. (Note: if ANSYS gives a default exterior angle instead of the interior angle, right click and select **Alternate Angle**.)



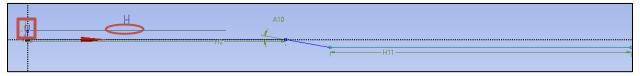


#### 4.12 Sketching > Draw > Line. Create a horizontal line as per below.

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

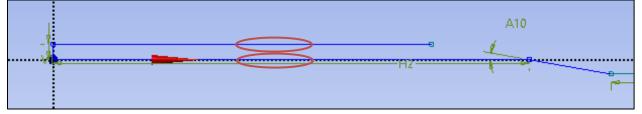


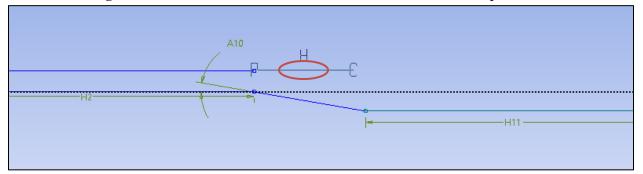
4.14 Sketching > Draw > Line. Draw the horizontal line circled in red line as per below.



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

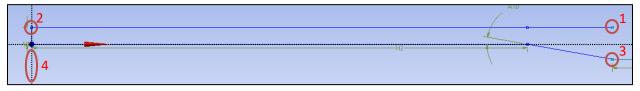
Sketching Toolboxes
Draw
Modify
Dimensions
Constraints
Trived Try Fixed Try Horizontal I Vertical ✓ Perpendicular ⊗ Tangent Coincident → Midpoint ↑ Symmetry Ø Parallel © Concentric
≫ Foual Radius ↓ È Equal Length
Equal Distance Auto Constraints
Settings 👻
Sketching



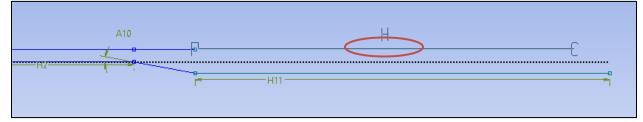


4.16 Sketching > Draw > Line. Draw the horizontal line circled in red as per below.

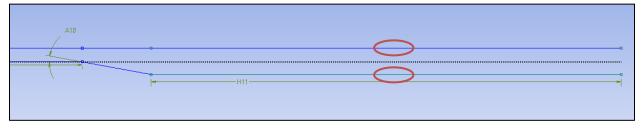
4.17 **Sketching** > **Constraints** > **Equal Distance**. Click on Point 1 and then click on the Point 2. Click Point 3 and then click on line 4. This makes points 1 and 3 the same distance from the y-axis in the horizontal direction.



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



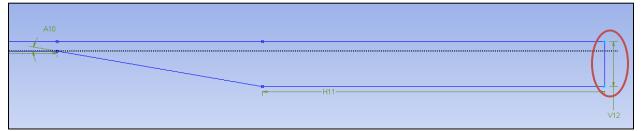
#### 4.19 Sketching > Constraints > Equal Length. Click on two lines circled in red as below.



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.

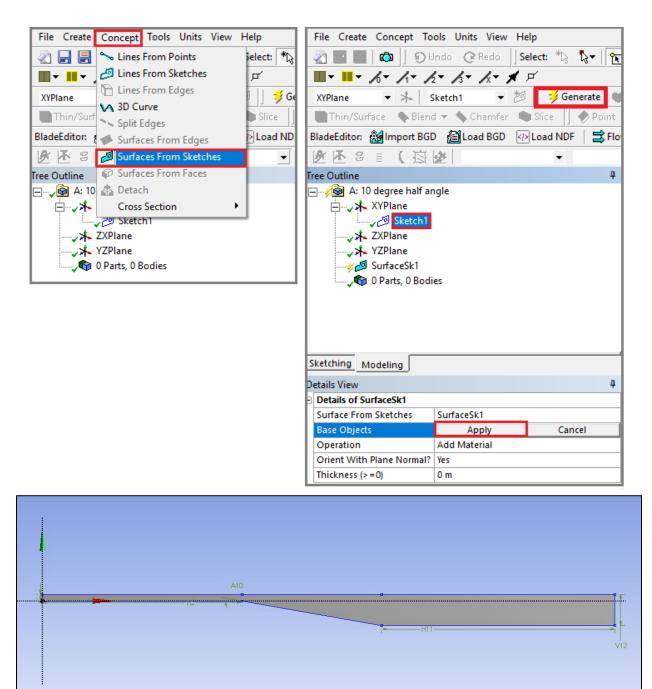
Н11-	A10	~	
------	-----	---	--

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.

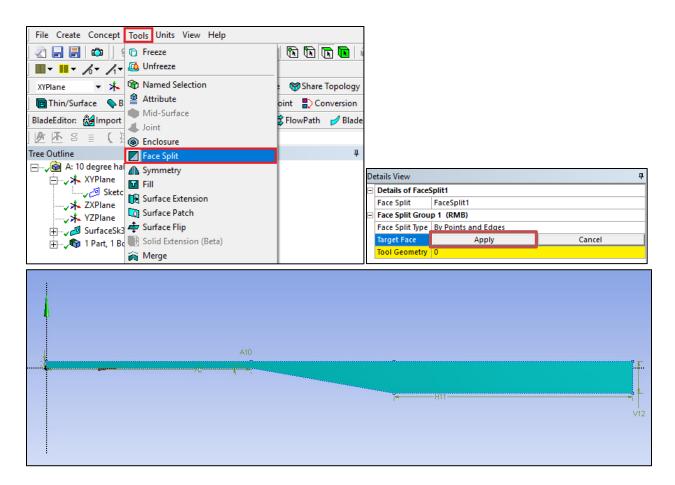


De	etails View	Ф
=	Details of Sketch1	
	Sketch	Sketch1
	Sketch Visibility	Show Sketch
	Show Constraints?	No
Ξ	Dimensions: 5	
	A3	10 °
	H2	60 m
	H4	70 m
	□ V1	2 m
	V5	9.4 m
Ξ	Edges: 8	
	Line	Ln7
	Line	Ln8
	Line	Ln9
	Line	Ln10
	Line	Ln11
	Line	Ln12
	Line	Ln13
	Line	Ln14

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

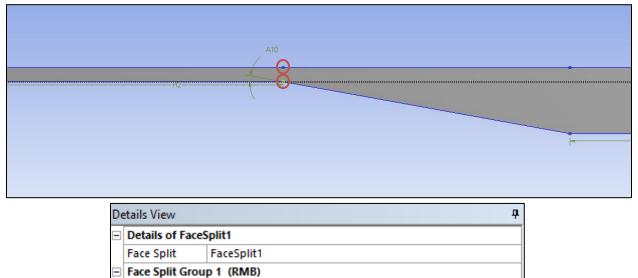


4.23 **Tools** > **Face Split**. Select the surface you created (it will be highlighted in green when you select it as shown below) then click **Apply** for **Target Face**.



4.24 Click on the yellow region shown below.

De	etails View	<b>ф</b>
-	Details of Faces	Split1
	Face Split	FaceSplit1
-	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

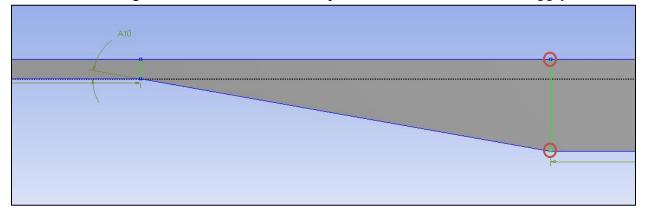
1

De	etails View	р
Ξ	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

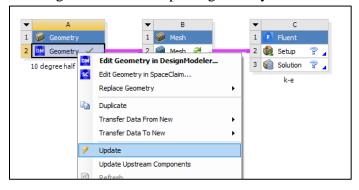
4.27 While holding **Ctrl** button click on the two points circled in red then click **Apply** button.



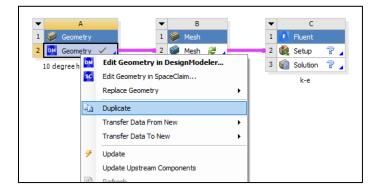
File Create Concept Tools	Units View	Help									
2 🔄 🔳 👛 🛛 💬 Und	lo @Redo	Select: *D	k- 🕞 🖬 🛛	3 <b>-</b> 1 - 1 - 1 - 1	(∭ S∳®	Q (H) (Q) (Q) (Q)	0, 🛱 🎁 🗼 🚳	• •			
I - I - h- h- h-	1- 1-	<b>1</b> 🖻									
XVPlane 🔹 👫 Ske	tch1 🔹	2 Gen	erate 🖤 Share	Topology 🔣 Para	meters 📗 🛄 Extra	ade 💼 Revolve 🐧	Sweep 💧 🐥 Skin/Loff				
Thin/Surface Slend >	🔹 🦘 Chamfer	Slice 4	👂 Point 🔹 🗋 Co	nversion							
BladeEditor: 👸 Import BGD	🛃 Load BGD	Icoad NDF	📑 FlowPath	🥩 Blade 🛛 💋 Spl	itter 🚽 Vista TFExp	ort 📉 ExportPoints	E StageFluidZone	🛃 SectorCut 🛛 🎪 Th	nroatArea 🛛 🧩 CAD Im	port 👻 🐼 Preference	s
逐还3 = ( 海陵				- 6	i 🖉 📾 🧭 👘						
Tree Outline		4	Graphics								
→ A 10 degree haf angle → A NVPlane → ZPlane → X V2Plane → X V2Plane → X V2Plane → ZPlane → ZPlane	e			<b>—</b> —н	6				12		
			1								
Sketching Modeling											
Details View Details of FaceSplit1		ą.									
Face Split FaceSplit1											
Face Split Group 1 (RMB)											
Face Split Type By Points and	Edges										Y
Target Face 1											<b></b>
Tool Geometry 2											•
											4
						0.00	30.00		60.00 (m)		-
							5.00	15.00			
							5.00	45.00			

4.28 Click the Generate button and Save your progress.

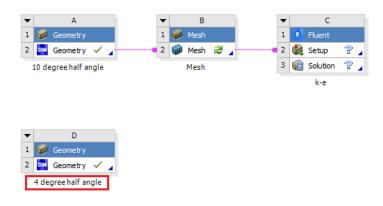
4.29 Close the ANSYS Design Modeler and update geometry



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.

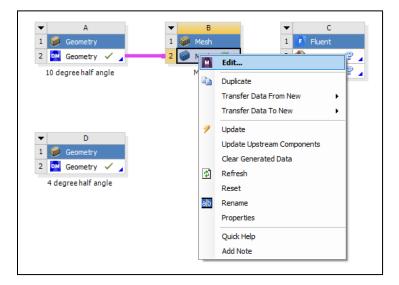
▼ A ▼ B		ketching Modeling	9
	1 Fluent 2 🍓 Setup 😨 🖌	etails View	
	3 🕼 Solution 😨	Sketch Visibility	Show Sketch
zo ogreenen ongre	k-e	Show Constraints?	No
	=	Dimensions: 5	·
		A3	4 °
▼ D		🗌 H4	70 m
1 🥪 Geometry		H6	60 m
2 Geomet Edit Geometry in DesignModeler		🗌 V1	2 m
4 degree half sc Edit Geometry in SpaceClaim		🗌 V5	9.4 m
Replace Geometry	=	Edges: 8	·
(b. a.t.)		Line	Ln7
Tree Outline		Line	Ln9
□		Line	Ln10
ZXPlane		Line	Ln11
YZPlane		Line	Ln12
⊟√2 SurfaceSk3		Line	Ln13
FaceSplit1		Line	Ln14
⊕, 🏟 1 Part, 1 Body		Line	Ln15

4.33 Save your file and quit ANSYS Design Modeler

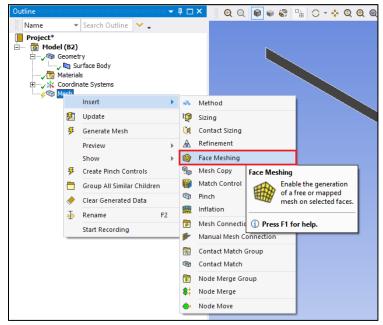
# 5. Mesh Generation

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...



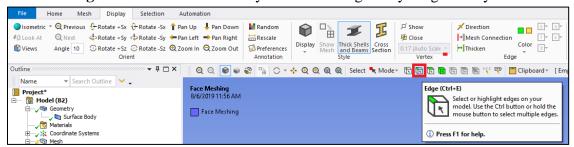
5.2 Right click on **Mesh** then select **Insert** > **Face Meshing**.



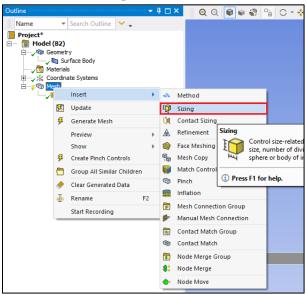
5.3 Select all three surface while holding **Ctrl** button and click **Apply** located Geometry.

De	etails of "Face Meshing" - Mappe	ed Face Meshing	
-	Scope		
	Scoping Method	Geometry Selection	
	Geometry	Apply	Cancel
Ξ	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**.



5.6 While holding Ctrl, click on the edges show	wn below and click <b>Apply</b> .
---	-----------------------------------

$\bigcirc$		(	)	
U				
	Details of "Edge Sizin	g" - Sizing	ņ	
E	Scope			
	Scoping Method	Geometry Selec	tion	
	Geometry	Apply	Cancel	
E	Definition			
	Suppressed	No		
	Туре	Element Size		
	Element Size	Element Size Default (2.3017 m)		
E	Advanced			
	Behavior	Soft		
	Growth Rate	Default (1.2)		
	Capture 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).

Ξ	Scope			
	Scoping Method	Geometry Selection		
	Geometry	2 Edges		
	Definition			
	Suppressed	No		
	Туре	Number of Divisions		
	Number of Divisions	59		ļ
	Advanced			
	Behavior	Hard		ļ
	Capture Curvature	No		ł
	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**.

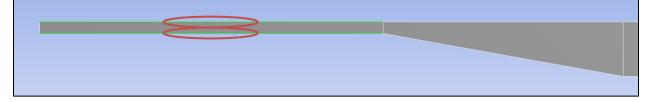


5.10 Change parameter for **Edge Sizing** as per below and click **Apply** (Right edge is shown as an example).

-	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	87.76

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

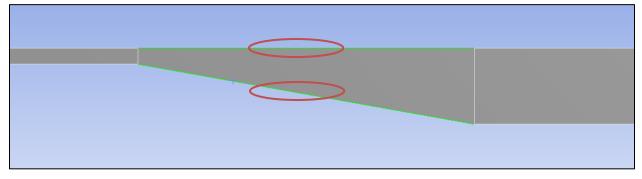
5.12 While holding **Ctrl**, click on the edge shown below and click **Apply**.



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

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		

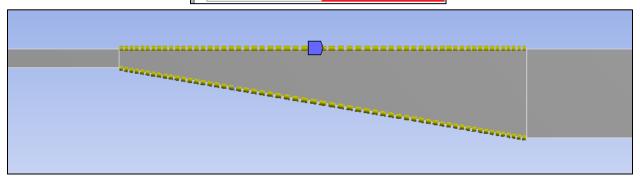
## 5.14 Right click on **Mesh** and **Insert** > **Sizing**.



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			
	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			



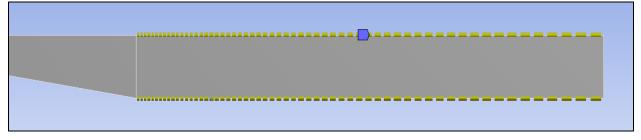
### 5.17 Right click on **Mesh** and **Insert** > **Sizing**.

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

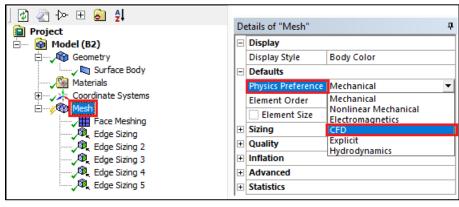


5.19 Change parameter for Edge Sizing as per below and click Apply.

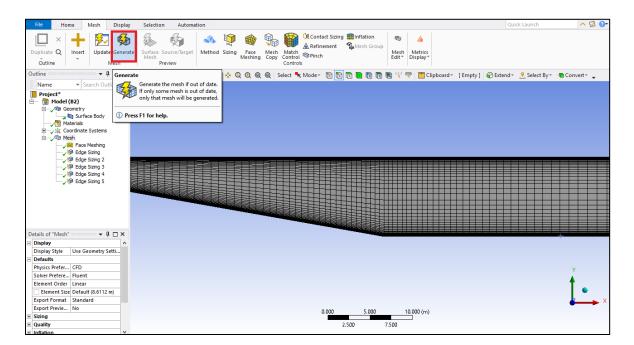
De	Details of "Edge Sizing 5" - Sizing 4				
	Scope				
	Scoping Method	Geometry Selection			
	Geometry	2 Edges			
	Definition				
	Suppressed	No			
	Туре	Number of Divisions			
	Number of Divisions	59			
	Advanced				
	Behavior	Hard			
	Capture Curvature	No			
	Capture Proximity	No			
	Bias Type				
	Bias Option	Bias Factor			
	Bias Factor	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.



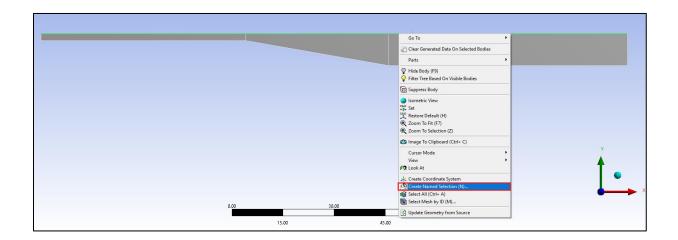
- Outline Name Search Outline Project\* 🖮 🐻 Mod<u>el (B2)</u> . Geometry 🙀 🔄 Surface Body 🖉 🗖 Materials 🗄 🧹 🔆 Coordinate Systems ⊡….,∕© Mesh 🏹 🎇 Face Meshing 🖓 🕼 Edge Sizing 🎣 🖗 Edge Sizing 2 √ 🕼 Edge Sizing 3 🏑 🕼 Edge Sizing 4 🏸 Edge Sizing 5 Home Geometry Display Selection Automation ► X Named Selection Images **≵**© Duplicate Q Comment 🔆 Coordinate System 🛄 Section Plane Annotation Geometry Virtual Body Virtual **→** ‡ 🗆 × Outline 🔅 . Q Q 📦 🗣 🐏 🔿 × أ Q Q Q Q Select 🥆 Mode × 🟗 🔃 🖪 🐘 🐘 🌾 🌱 🛅 Clipboard × [ ▼ Search Outline V 🗸 Name Edge (Ctrl+E) Geometry 8/6/2019 12:07 PM Project\*
- 5.22 Select **Geometry** to hide the mesh and click the **Edge** button.

👩 Model (B2)

E-Geometry

Vaterials

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)*.



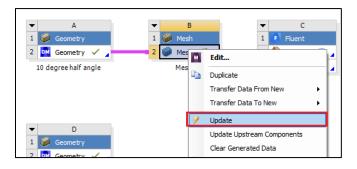
Select or highlight edges on your model. Use the Ctrl button or hold the

mouse button to select multiple edges.

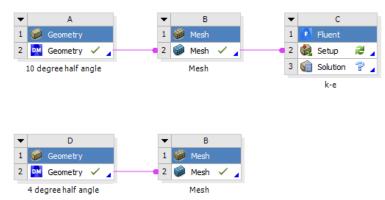
(i) Press F1 for help

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



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

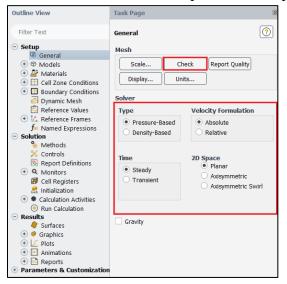


# 6. Setup

6.1 Right click **Setup** and click **Edit.** 

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

59490.0424330	of ANS	YS
er Readi	ing	
s panel a	gain	
chine)		
	1	\$
Machine	0	\$
	er Readi panel a : <b>hine)</b>	er Reading panel again thine)



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
	Cmu
🔿 Laminar	0.09
<ul> <li>Spalart-Allmaras (1 eqn)</li> </ul>	C1-Epsilon
k-epsilon (2 eqn)	1.44
🔿 k-omega (2 eqn)	C2-Epsilon
<ul> <li>Transition k-kl-omega (3 eqn)</li> </ul>	1.92
<ul> <li>Transition SST (4 eqn)</li> </ul>	TKE Prandtl Number
O Reynolds Stress (5 eqn)	1
Scale-Adaptive Simulation (SAS)	TDR Prandtl Number
<ul> <li>Detached Eddy Simulation (DES)</li> </ul>	1.3
k-epsilon Model	
Standard	
O Realizable	
Near-Wall Treatment	User-Defined Functions
Standard Wall Functions	Turbulent Viscosity
Scalable Wall Functions	
O Non-Equilibrium Wall Functions	Prandtl Numbers
Enhanced Wall Treatment	TKE Prandtl Number
O Menter-Lechner	none
O User-Defined Wall Functions	TDR Prandtl Number
	none
Enhanced Wall Treatment Options	
Pressure Gradient Effects	
Options	
Curvature Correction	
Production Kato-Launder	
Production Limiter	
ОК Сал	cel Help

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

Create/Edit Materials				×
Name air Chemical Formula		Material Type fluid Fluent Fluid Materials air Mixture none	• •	Order Materials by <ul> <li>Name</li> <li>Chemical Formula</li> </ul> Fluent Database User-Defined Database
	Properties Density (kg/m3) Viscosity (kg/m-s)	1	Edit	
	Change	e/Create Delete Close Help		

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 l	nlet						×
Zone Name							
inlet							
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
	Velocity Spe	ecification Me	ethod Comp	onents			•
		Reference F	rame Absolu	ıte			-
Supersonic/	Initial Gauge	Pressure (pa	iscal) 0				•
		X-Velocity	(m/s) 1.25				~
		Y-Velocity	(m/s) 0				•
Turbu	lence						
	Spec	cification Met	hod K and E	psilon			•
Tur	bulent Kinetic	: Energy (m2	/s2) 0.0018				-
Turbulent Dissipation Rate (m2/s3) 0.0000963						-	
			OK Can	cel Help			

Inlet Boundary Condition						
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         Momentum       Thermal       Radiation       Species       DPM       Multiphase       Potential         Backflow Reference Frame       Absolute       Gauge Pressure (pascal) 0       Pressure Profile Multiplier 1         Backflow Direction Specification Method       Normal to Boundary       Backflow Pressure Specification Total Pressure         Average Pressure Specification       Target Mass Flow Rate       Turbulence         Specification Method       Intensity and Length Scale       Backflow Turbulent Intensity (%) 3.25	
Backflow Reference Frame Absolute Gauge Pressure (pascal) Pressure Profile Multiplier 1 Backflow Direction Specification Method Normal to Boundary Backflow Pressure Specification Total Pressure Average Pressure Specification Target Mass Flow Rate Turbulence Specification Method Intensity and Length Scale	
Gauge Pressure (pascal) 0 Pressure Profile Multiplier 1 Backflow Direction Specification Method Normal to Boundary Backflow Pressure Specification Total Pressure Average Pressure Specification Target Mass Flow Rate Turbulence Specification Method Intensity and Length Scale	UDS
Pressure Profile Multiplier 1 Backflow Direction Specification Method Normal to Boundary Backflow Pressure Specification Total Pressure Average Pressure Specification Target Mass Flow Rate Turbulence Specification Method Intensity and Length Scale	
Backflow Direction Specification Method Normal to Boundary Backflow Pressure Specification Total Pressure Average Pressure Specification Target Mass Flow Rate Turbulence Specification Method Intensity and Length Scale	
Backflow Pressure Specification Total Pressure Average Pressure Specification Target Mass Flow Rate Turbulence Specification Method Intensity and Length Scale	
Average Pressure Specification Target Mass Flow Rate Turbulence Specification Method Intensity and Length Scale	
Target Mass Flow Rate Turbulence Specification Method Intensity and Length Scale	
Turbulence Specification Method Intensity and Length Scale	
Specification Method Intensity and Length Scale	
Backflow Turbulent Intensity (%) 3.25	-
	-
Backflow Turbulent Length Scale (m) 0.0035	-

Outlet Boundary Condition							
Variable	u (m/s)	v (m/s)	P (Pa)	Intensity (%)	Length scale (m)		
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 💫 📻	Zone Filter Text bottom_wall interior-surface_body outlet surface_body top_wall
Phase Type D mbture Wall Wall W 6 Edit Copy Profiles Parameters Display Mesh Periodic Conditions	Phase Type D midure Vall V Edit Copy Profiles Parameters Display Mesh Periodic Conditions

Wall Boundary Condition						
Variable	u (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	(?)
<ul> <li>Setup</li> <li>General</li> </ul>	Compute from	•
<ul> <li>Image: Image: Im</li></ul>	Reference Values	
🕑 🖽 Cell Zone Conditions	Area (m2) 0.25	
Boundary Conditions	Density (kg/m3) 1	
🖉 Dynamic Mesh 🔁 Reference Values	Depth (m) 1	
Kererence Values	Enthalpy (j/kg) 0	
for Named Expressions	Length (m) 1	
<ul> <li>Solution</li> <li>Methods</li> </ul>	Pressure (pascal) 0	
	Temperature (k) 288.16	
Report Definitions	Velocity (m/s) 1.25	
📀 🔍 Monitors	Viscosity (kg/m-s) 0.000147	
Cell Registers	Ratio of Specific Heats 1.4	1
<ul> <li>Initialization</li> <li>Calculation Activities</li> </ul>		
Calculation Activities	Reference Zone	
<ul> <li>Results</li> </ul>		<b></b>
Surfaces		
📀 🔮 Graphics		

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

In case of 'Yplus for Heat Tran. Coef' 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
<ul> <li>Setup         <ul> <li>General</li> <li>General</li> <li>Models</li> <li>Materials</li> <li>Cell Zone Conditions</li> <li>Boundary Conditions</li> <li>Dynamic Mesh</li> <li>Reference Values</li> <li>X. Reference Frames</li> <li>Named Expressions</li> </ul> </li> <li>Solution         <ul> <li>Methods</li> <li>Controls</li> <li>Report Definitions</li> <li>Monitors</li> <li>Cell Registers</li> <li>Initialization</li> <li>Calculation Activities</li> </ul> </li> </ul>	Solution Methods         Pressure-Velocity Coupling         Scheme         SIMPLE         Spatial Discretization         Gradient         Green-Gauss Cell Based         Pressure         Second Order         Momentum         Second Order Upwind         Turbulent Kinetic Energy         Second Order Upwind         Turbulent Kinetic Energy         Second Order Upwind         Turbulent Dissipation Rate         Second Order Upwind
<ul> <li>Event Calculation</li> <li>Results</li> <li>Surfaces</li> <li>Graphics</li> </ul>	Transient Formulation
(*) 🗾 Plots	Non-Iterative Time Advancement

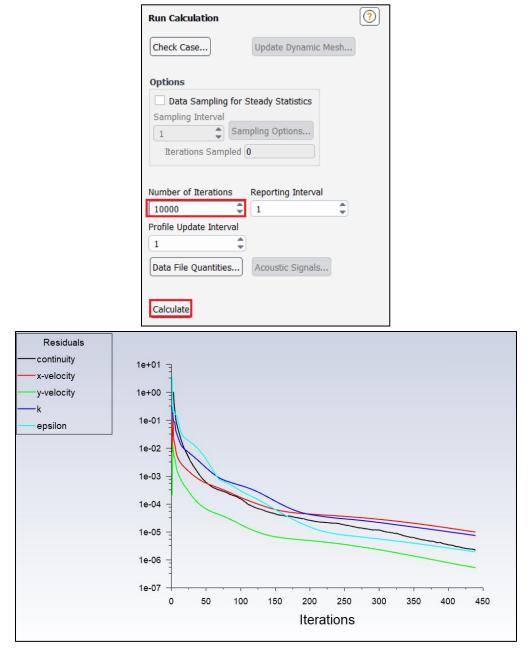
6.11 **Tree > Solution > Monitors > Residual.** Change convergence criterions to 1e-05 and click **OK(Apply)**.

Residual Monitors					×
Options	Equations				
✓ Print to Console	Residual	Monitor	Check Convergence	e Absolute Criteria	
✓ Plot	continuity	<ul> <li>Image: A start of the start of</li></ul>	✓	1e-05	
Window	x-velocity		$\checkmark$	1e-05	
1 Curves Axes	y-velocity	<ul><li>✓</li></ul>	$\checkmark$	1e-05	
Iterations to Plot	k	<ul><li>✓</li></ul>	$\checkmark$	1e-05	
1000	epsilon	<b>v</b>	✓	1e-05	
Iterations to Store	Residual Values		Conver	gence Criterion	
	Normalize	Itera	tions	ite	*
		5	*		
	✓ Scale		Conve	ergence Condition	ıs
	Compute Local S	Scale			
ОКР	lot	Cancel	Help		

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

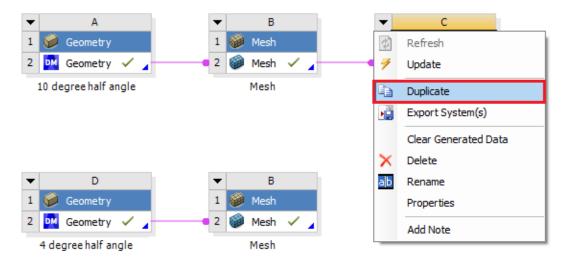
Solution Initialization	?
Initialization Methods	
<ul><li>Hybrid Initialization</li><li>Standard Initialization</li></ul>	
Compute from	
<b>•</b>	
Reference Frame	
Relative to Cell Zone     Absolute	
Initial Values	
Gauge Pressure (pascal)	
0	
X Velocity (m/s)	
0.887	
Y Velocity (m/s)	
0	
Turbulent Kinetic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

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

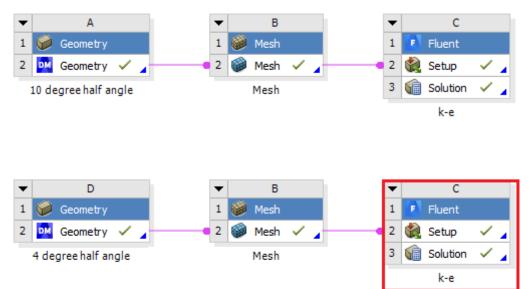


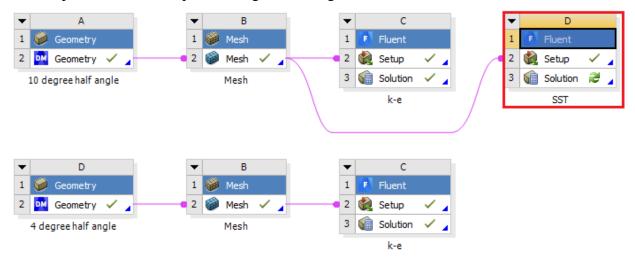
6.14 Save your project and quit ANSYS fluent.

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.



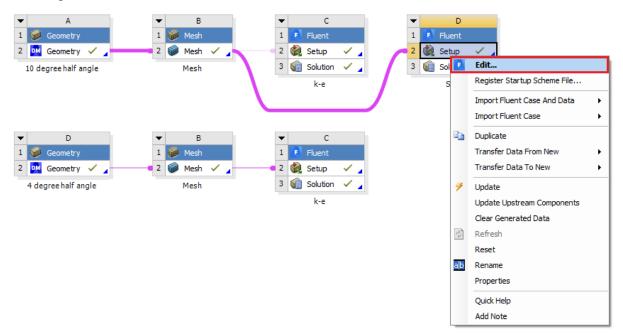
After simulation runs





#### 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	
<ul> <li>Inviscid</li> <li>Laminar</li> <li>Spalart-Allmaras (1 eqn)</li> <li>k-epsilon (2 eqn)</li> <li>k-omega (2 eqn)</li> <li>Transition k-kl-omega (3 eqn)</li> </ul>	Alpha*_inf 1 Alpha_inf 0.52 Beta*_inf	
Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) Transition SST Options Roughness Correlation	0.09 a1 0.31 Beta_i (Inner) 0.075 Beta_i (Outer)	
Options	User-Defined Transition Correlations	
	F_length	

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

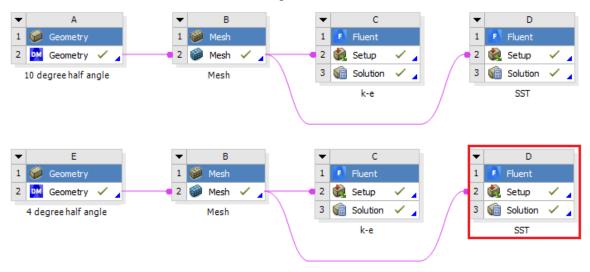
Solution Controls	?
Under-Relaxation Factors Density	
1	
Body Forces	
1	
Momentum	
0.5	
Turbulent Kinetic Energy	
0.5	
Specific Dissipation Rate	
0.5	
Intermittency	
0.5	
Momentum Thickness Re	
0.5	
Turbulent Viscosity	
1	
	<b>~</b>
Default	
Equations Limits Advanced	d)

## 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 Design	1 🔺	
Display     Info + Operation of the second	Scale	Zones Separate → Delete Separate → Delete Adjacency  Adjacency	Image: Append     ↓       Image: Append image: Appe	Interfaces Mesh Overset	Mesh Models Dynamic Mesh Mixing Planes Turbo Topology	Adapt Refine / Coarsen	Surface + Create - Zone Partition
Outline View	Task Page Run Calculation						Imprint Point Line/Rake
<ul> <li>Setup</li> <li>I General</li> <li>I Models</li> <li>I Materials</li> <li>I Cell Zone Conditions</li> <li>I Boundary Conditions</li> </ul>	Check Case U Options Data Sampling for Ster	pdate Dynamic Mesh					Plane. Line/Rake Quadric Iso-Surface Iso-Clip
Ø Dynamic Mesh B Reference Values	Sampling Interval	ng Options					Transform

📧 Line/Rake Su	rface		×
New Surface Na	me		
position-1			
Options			Number of Points
Line	Туре		10
Reset	Line	•	)
			•
End Points			
x0 (m) 78		x1 (m) 78	
y0 (m) -3.52		y1 (m) 2	
z0 (m) 0		z1 (m) 0	
	Select Point	ts with Mou	se
	Create	Close	р

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	Soluti	on Re	esults
Field Func	tions	User I	Defined			Model Specif	ic
Custom		🐔 Eunction Ho		💾 Mer	nory	😑 1D Coupling	g
🖋 Units		Tel Function Ho	00KS	X Sca	lars	🕙 Fan Model.	
िन्न Parame	t Custom Create custo	m field functions	Demand	📑 Rea	d Table		

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

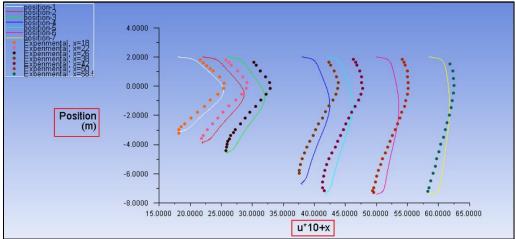
7.3 Plotting modified U and modified TKE

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

	-	ions as shown below. Turn on "Position on Y Axis"
E Solution XY Plot		×
XY Plot Name		
xy-plot-1		
Options	Plot Direction	Y Axis Function
✓ Node Values	X 0	Direction Vector
Position on X Axis	Y 1	X Axis Function
Position on Y Axis	Z (0	Custom Field Functions
Write to File		u*10+x
Order Points		u 10+x
File Data [1/1] = = = = = = = = = = = = = = = = = = =	Load File Free Data	Surfaces Filter Text To Text Text Text Text Text Text Text Text
Save/Plot	Axes Curve	es Close Help

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



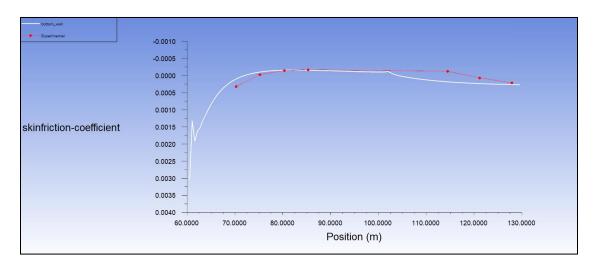
7.4 Plotting skin friction coefficient

**Results** > **Plots** > **XY Plot** > **Load File...** Select the 'Skin\_Friction\_bot\_wall.xy' file downloaded from the class website and click **OK**.

enunge me paran		
Solution XY Plot		×
XY Plot Name		
xy-plot-2		
Options	Plot Direction	Y Axis Function
✓ Node Values	X 1	Custom Field Functions
Position on X Axis	YO	skinfriction-coefficient
Position on Y Axis	Z 0	X Axis Function
Write to File		Direction Vector
Order Points		
File Data [1/1]		Surfaces (Filter Text 💦 🔂 🛃 🛃
File Data [1/1]	Load File	📀 Inlet
Skin friction. Lower wall (H:/9. TA/Skin_frict		Internal
	Free Data	Line-surface     Outlet
		<ul> <li>Wall</li> </ul>
		bottom_wall
		top_wall
		New Surface
		new surface *
Save/Plot	Axes Curve	es 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

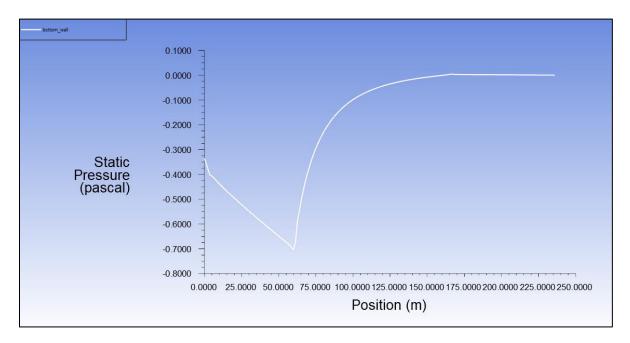
Force Reports			×
Options  Forces Moments Center of Pressure Save Output Parameter	Direction Vector	Wall Zones Filter Text bottom_wall top_wall	
	Print	te Close Help	2

Forces Zone bottom_wall	Forces (n) Pressure (1.058987 38.	917494 0)		Viscous (0.34016388 -0.0	0010073304 0)		Total (1.3991509 38.916487 0)
Net	(1.058987 38.	917494 0)		(0.34016388 -0.0	010073304 0)		(1.3991509 38.916487 0)
Forces - Direction Vector	(1 0 0) Forces (n)			Coefficients			
Zone bottom_wall	Pressure 1.058987	Viscous 0.34016388	Total 1.3991509	Pressure 5.4220134	Viscous 1.7416391	Total 7.1636525	
Net	1.058987	0.34016388	1.3991509	5.4220134	1.7416391	7.1636525	

7.6 Finding the pressure difference between inlet and outlet.

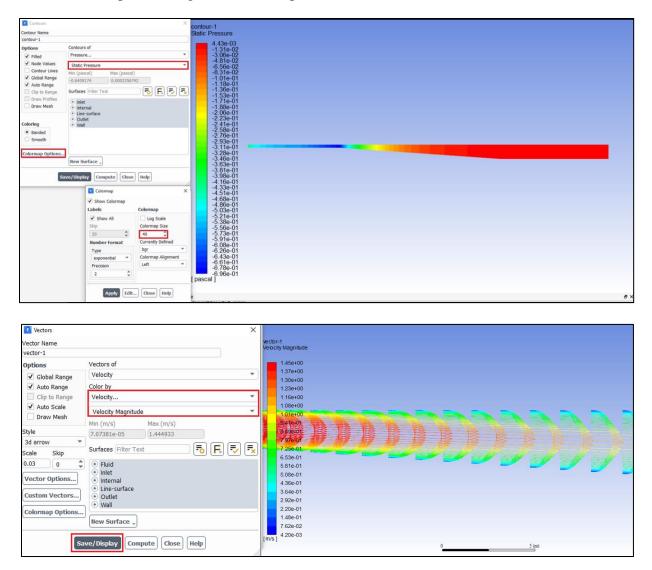
You can simply write pressure at bottom wall to a file and take the difference of pressure at inlet and outlet.

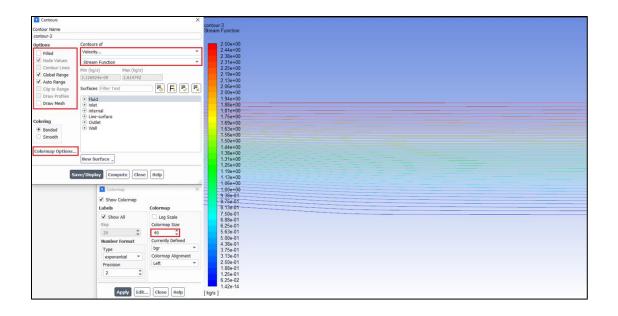
Solution XY Plot			×
XY Plot Name			
xy-plot-3			
Options	Plot Direction	Y Axis Function	
✓ Node Values	X 1	Pressure 🔻	
Position on X Axis	Y 0	Static Pressure 💌	
Position on Y Axis	Z 0	X Axis Function	
✓ Write to File		Direction Vector	
Order Points		Surfaces Filter Text	) (= <u>×</u>
File Data [0/0]	Load File	📀 Inlet	
	Load Them	💽 Internal	
	Free Data	Line-surface	
		Outlet     Wall	
		bottom_wall	
		top_wall	
		New Surface 🚽	
	Write Axes Curves	s) Close Help	



7.7 Plotting contours, velocity vectors and streamlines.

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.





# 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			
<b>Relative error (%)</b>			

- 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		
<b>Relative error (%)</b>		

- 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-  $\varepsilon$  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 <b>←Section 8 (Page# 47) for CFD Lab 3</b>		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