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

Verification and Validation of Turbulent Flow around a Clark-Y Airfoil

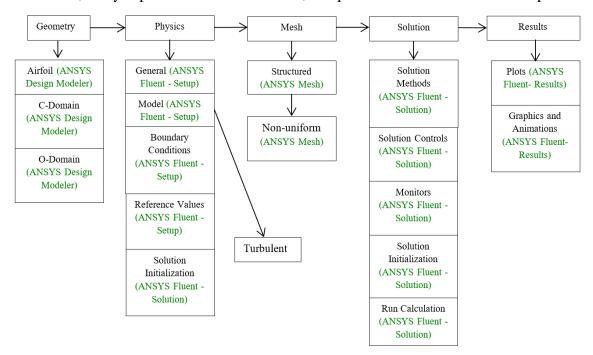
ME:5160 Intermediate Mechanics of Fluids CFD LAB 2 (ANSYS 2023 R1; Last Updated: August 16, 2023)

By Timur Dogan, Michael Conger, Dong-Hwan Kim, 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 2 is to simulate turbulent airfoil flows following "CFD process" by an interactive step-by-step approach and to conduct verifications. Students will have "hands-on" experiences using ANSYS to conduct verification for lift coefficient and pressure coefficient distributions, and validation for pressure coefficient distribution, including effect of numerical scheme. Students will manually generate C type mesh and investigate the effect of domain size and effect of angle of attack on simulation results. Students will analyze the differences between CFD and EFD, analyze possible sources of errors, and present results in a CFD Lab report.



Flow Chart for "CFD Process" for airfoil flow

2. Simulation Design

The problem to be solved is that of turbulent flows around a Clark-Y airfoil. Reynolds number is 143,000 based on the inlet velocity and airfoil chord length. The following figures show the illustrations for C type and O type domains (Note: the figures are not in the exact scale as the true size of the domain and airfoil).

Table 1 - Main particulars									
Parameter	Symbol	Unit	O-type	C-Type					
Chord Length	С	m	0.3048	0.3048					
Downstream length	Lo	m	-	5					
Radius	Rc	m	5,4,3,2,1	5					
Angle of attack	α	degree	0,6	0					

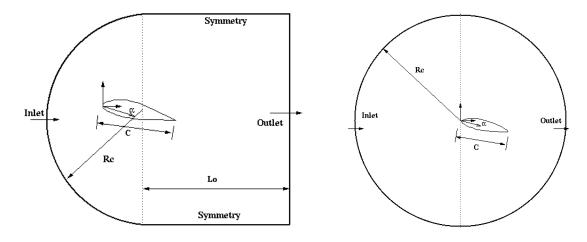


Figure 1 - C and O domain shapes and boundary conditions

In CFD Lab 2, boundary conditions for C type of mesh will be "inlet", "outlet", "symmetry" and "airfoil", as described later. Boundary conditions for O type of meshes will be "inlet", "outlet", and "airfoil". Uniform flow was specified at inlet. For outlet, zero gradients are fixed for all velocities and pressure is constant. No-slip boundary condition will be used on the "airfoil". Symmetric boundary condition will be applied on the "symmetry". The meshes and the simulations that will be conducted are shown in Tables 2 and 3 respectively.

Table 2 - Mesh										
Mesh Name	Domain Type	Radius [m]	Angle of Attack (AOA) [degree]							
C-mesh	С									
fine										
medium		5								
coarse										
Domain-R5			0							
Domain-R4	0	4								
Domain-R3		3								
Domain-R2		2								
Domain-R1]	1								
AOA6		5	6							

Table 2 - Mesh

Table 3	- Simulation	Matrix
I abie e	omunation	TARGET 121

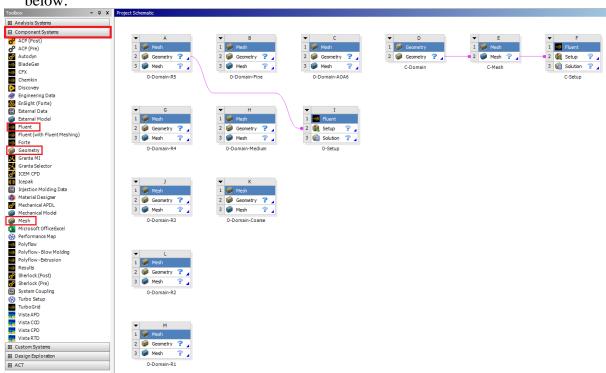
Study	Mesh
Domain size	Domain-R1, Domain-R2, Domain-R3, Domain-R4, Domain-R5
V&V and effect of numerical scheme	fine, medium, coarse
Domain shape	C-mesh
Angle of attack	AOA6

All EFD data and CFD materials for turbulent airfoil flow in this Lab can be downloaded from class website (<u>http://www.engineering.uiowa.edu/~me_160/</u>).

3. ANSYS Workbench

3.1. Start > All Programs > ANSYS 2022 R1> Workbench 2022 R1

3.2. Toolbox > **Component Systems**. Drag and drop **Geometry**, **Mesh** and **Fluent** components to **Project Schematic** and name components as per below (Only C-Domain requires **Geometry**; i.e. other cases directly use **Mesh** skipping **Geometry**). Create connection as per below.



3.3. File > **Save As...** Save project on the network drive so that you can access anywhere in Engineering building (recommended). Name the file "CFD Lab 2".

4. Geometry

DesignModeler Geometry...

D E F 1 😥 Mesh 1 Fluent ometry 2 🎡 Setup 7 New SpaceClaim Geometry... 3 👔 Solution ? New DesignModeler Geometry... C-Setup New Discovery Geometry... D Import Geometry ۲ Þ Duplicate Transfer Data From New ۲ Transfer Data To New ۲

4.1. From the Project Schematic, right click on the C-Domain Geometry and select New

- **4.2.** Make sure that Unit is set to **Meter** (default value).
 - D: C-Domain DesignModeler File Create Concept Tools Units View Help Meter **E E E** 🖉 📕 📕 🖾 🗌 💬 Undq ~ Centimeter **■ • / • / • / • / •** Millimeter XYPlane 🝷 📥 🛛 None 🖤 Share Topolo Micrometer Thin/Surface 🔷 🗣 Blend 🔻 nt 🚽 🔁 Conversio Foot FlowPath 🛛 🥑 Bla BladeEditor: 💏 Import BGD 🚦 Inch **A A** 3 (宮屋 Large Model Support ۲ Tree Outline ✓ Degree 🖃 🗸 👰 D: C-Domain Radian 🗸 🛧 XYPlane 🎝 🛧 ZXPlane Model Tolerance ۲ 🗸 📥 YZPlane 👘 0 Parts, 0 Bodies
- **4.3.** Go to the class website and download Airfoil geometry (You can download the file by right clicking and selecting **Save as** or **Save target as**).

4.4. File > **Import External Geometry File...** Select **airfoil.igs** downloaded and click **Open**. Click **Generate**. Please remind that view can be moved to xy-plane view by clicking z-axis on 3-D axis at right bottom of the window. Drag and drop with right mouse button only for zooming in. Use wheel to zoom in/out. Press F7 to restore the view.

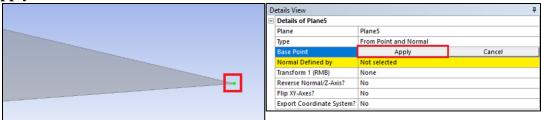
📴 D: C-Domain - DesignModeler	😡 D: C-Domain - DesignModeler
File Create Concept Tools Units View Help	File Create Concept Tools Units View Help
Refresh Input	😧 🔤 📕 🖚 🗍 💬 Undo 📿 Redo 🛛 Select: 🏗 🖏 🔭 🕞
Start Over (Ctrl+ N)	■ • ■ • <i>ト</i> • <i>ト</i> • <i>ト</i> • <i>ト</i> • <i>K</i> • <i>K</i>
🗌 🚰 Load DesignModeler Database (Ctrl+ O)	🛛 🔽 🖈 None 👻 📁 🧚 Generate 🖤 Shar
Save Project (Ctrl+ S)	Thin/Surface 💊 Blend 🔻 💊 Chamfer 🛸 Slice 📗 🛷 Point 🜓 0
Export	BladeEditor: 🆓 Import BGD 🕼 Load BGD 🐶 Load NDF 📑 FlowPati
Attach to Active CAD Geometry	
Import External Geometry File	
Import Shaft Geometry	
🗣 Write Script: Sketch(es) of Active Plane	
😪 Run Script	
Print 🖉	
🔂 Auto-save Now	
Restore Auto-save File	
Close DesignModeler	

4.5. Add a new plane by selecting the **New Plane (not New Sketch)** button. For the **Type** select **From Point and Normal**.

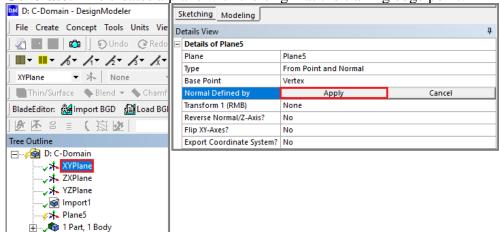
D: C-Domain - DesignModeler
File Create Concept Tools Units View Help
🖉 🔚 📕 📫 🗍 DUndo 🕜 Redo 🗍 Select: 🍢 🥾 🖻 🖻 🖻 🖻 🖉 🖉 🗐 🗮 🗒 🖉 🖉 🔍 🔍 🍭
■
XYPlane 🔻 🗚 None 👻 🗍 🏓 Generate 🍘 Share Topology 😰 Parameters 🗍 🖪 Extrude 🏘 Re
📴 Thin/Surface 🛛 💊 New Plane Chamfer 🏘 Slice 🖉 🄗 Point 📳 Conversion
BladeEditor: 🍰 Import BGD / 🗃 Load BGD / Doad NDF 🛛 😅 FlowPath 🥜 Blade 🚀 Splitter 🚽 Vista TFExport 📐 Exp
慶丞ऽ≡(哀陵│
ree Outline 4 Graphics
∃…, 👰 D: C-Domain

S	Sketching Modeling							
D	etails View		ф.					
=	Details of Plane5							
	Plane	Plane5						
	Туре	From Point and Normal						
	Base Point	Not selected						
	Normal Defined by	Not selected						
	Transform 1 (RMB)	None						
	Reverse Normal/Z-Axis?	No						
	Flip XY-Axes?	No						
	Export Coordinate System?	No						

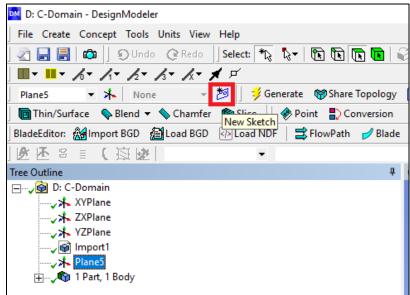
4.6. For the **Base Point**, zoom in and select the point at the trailing edge as seen below and click **Apply**.



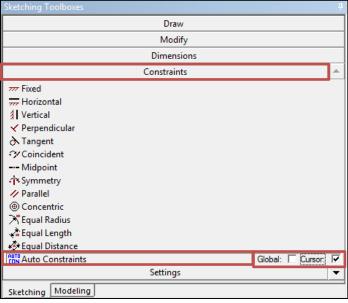
4.7. For the **Normal Defined By**, select the **XYPlane** on the **Tree Outline** and click **Apply**. Then click **Generate**. This creates a plane with the origin at the trailing edge point.



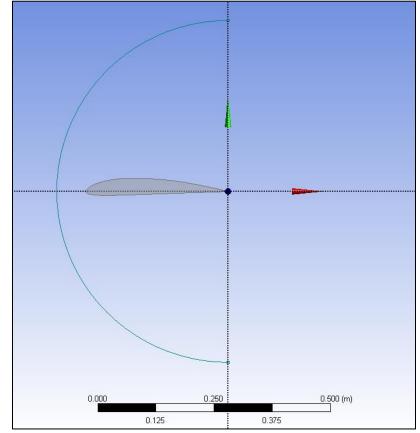
4.8. Make sure the plane you just created is selected under tree outline then click the **New Sketch** button.



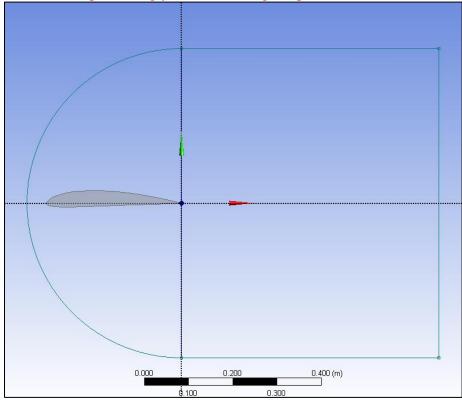
4.9. In **Sketching Toolboxes>Constraints**, Enable the **Auto Constraints** option to pick the exact point as below



4.10. Sketching > Draw > Arc by Center. Draw an arc centered at the trailing edge origin as per below. Make sure the end points are on the y-axis. The Auto Constraint will show "P" on the mouse point when it goes near the origin and "C" near the y-axis.



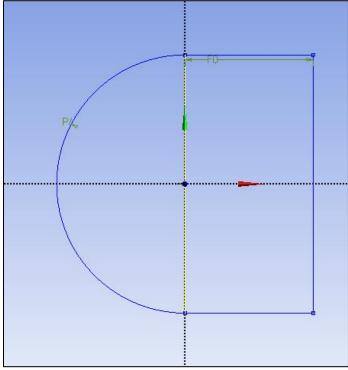
4.11. Sketching > Draw > Rectangle by 3 Points. Draw a rectangle as per below. Start by selecting one of the arc's ends and then select the other arc end, then pull the rectangle out to the right so it looks like the figure below and left click. Make sure, when selecting the arc ends, the "P" shows up ensuring you are selecting the point at the end of the arc.



4.12. Dimensions > **General**. Size the arc and rectangle with a radius of 5m and a width of 5m respectively as seen below (Do not add unit [m] when you put in the values). To use the tool, left click on the edge you want and click somewhere outside from the edge.

Details View			
Details of Sketch1			
Sketch	Sketch1	The	
Sketch Visibility	Show Sketch		
Show Constraints?	No		
Dimensions: 2			
H2	5 m		•
🗌 R1	5 m		
Edges: 5			
Circular Arc	Cr9		
Line	Ln10		
Line	Ln11		
Line	Ln12	0.000	4.000 8.000 (m)
Line	Ln13	2.000	6.000

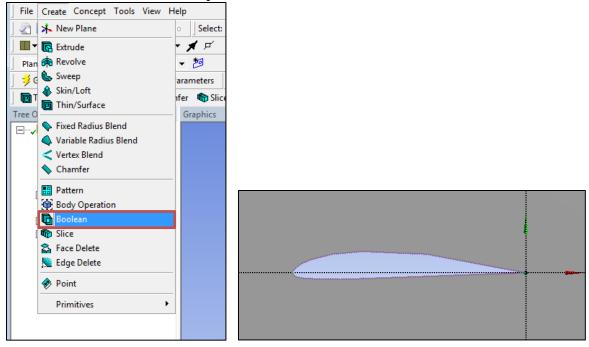
4.13. Delete the line that makes the left side of the rectangle by highlighting as yellow by selecting it and pressing **Delete** on the keyboard (This works only when "Sketching" tab is activated).



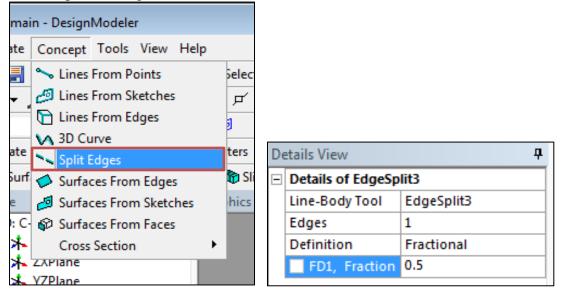
4.14. Concept > Surfaces from Sketches. Select the sketch you just made under tree outline, click Apply then click Generate.

File Create	Concept Tools Units View H	Help		
2 🖬 🖪	∾ Lines From Points	select: 🆎 💱 💽 💽 💽		F0
🔲 • 📕 • .	Lines From Sketches	д		
Plane5	The Lines From Edges	🕽 📔 🥩 Generate 🛛 🗑 Share Topolo		
Thin/Surf	Split Edges	🗑 Slice 🛛 🚸 Point 📳 Conversio	P/	
BladeEditor:	Surfaces From Edges	Eload NDF 🛛 😫 FlowPath 🥑 Bla		
	🖉 Surfaces From Sketches	•		
Tree Outline	🖗 Surfaces From Faces			
	🖓 Detach			
~×	Cross Section			
	ZXPlane	-		
	YZPlane			
⋳⊸≁	رضا Sketch1			
	SurfaceSk1			
	2 Parts, 2 Bodies			

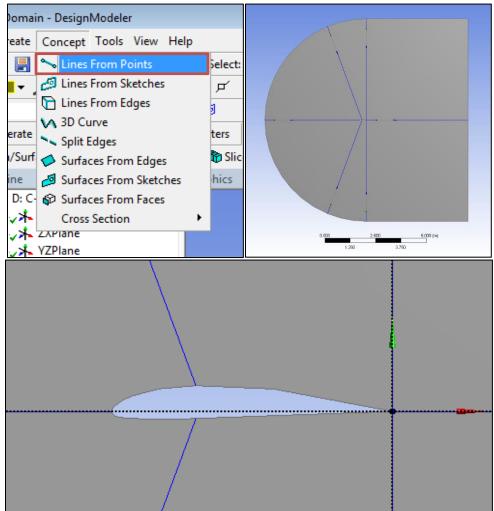
4.15. Create > Boolean. Make sure the Operation is set to Subtract. For the Target Body select the gray domain and click Apply. For the Tool Bodies select the airfoil by selecting the first Surface Body under the Tree Outline which corresponds to the airfoil (or simply zoom in near to the airfoil and select it) and click Apply. Then click Generate. This step will make the domain hollow as airfoil shape.



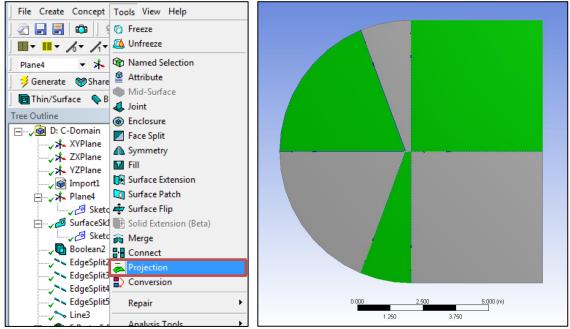
4.16. Concept > Split Edges. Select the arc and click Apply. Make sure the Fraction is set to 0.5. This splits the edge in half. Click Generate.



- **4.17.** Select the upper half of the arc you just split. Concept > Split Edges. Click Apply and change the Fraction to 0.25. This splits the top arc into two parts with the small piece towards the top of the screen. Click Generate.
- **4.18.** Repeat step 4.16 for the bottom piece of the arc you originally split but this time change the **Fraction** to 0.75. This will split the arc with the smaller piece towards the bottom of the screen. Click **Generate**.
- **4.19.** Split the vertical line from the rectangle in half as well. Make sure that **Fraction** is set to 0.5. Concept > Split Edges. Select the vertical line and click Apply. Click Generate.
- **4.20.** Concept > Lines From Points draw lines from the domain perimeter to the perimeter of the airfoil always starting from the domain and ending at the airfoil. Only for the line going from the trailing edge of the airfoil to the domain, start from the airfoil. Make sure the arrows have the same direction as shown in the below figure. Do this by selecting the point on the domain, hold Ctrl and select the point on the airfoil. Click Apply and then Generate. Repeat this process to create all the lines shown below. NOTE: If you do not create your lines starting from domain and ending in airfoil you will need to use a different bias type in the Mesh Generation.



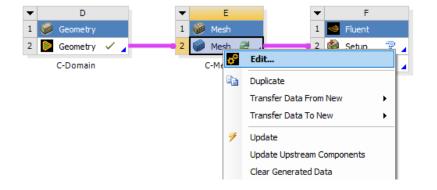
4.21. Tools > Projection. For the Edges select all the lines you just created by holding Ctrl while selecting them and then click Apply. For the Target select the surface of the domain and click Apply. Click Generate. This splits the domain into six sections as seen below.



4.22. File > Save Project. Close window.

5. Mesh

In this section C-Mesh will be manually generated and the O-type Meshes will be imported.



5.1. From the Project Schematic right click on C-Mesh and select Edit.

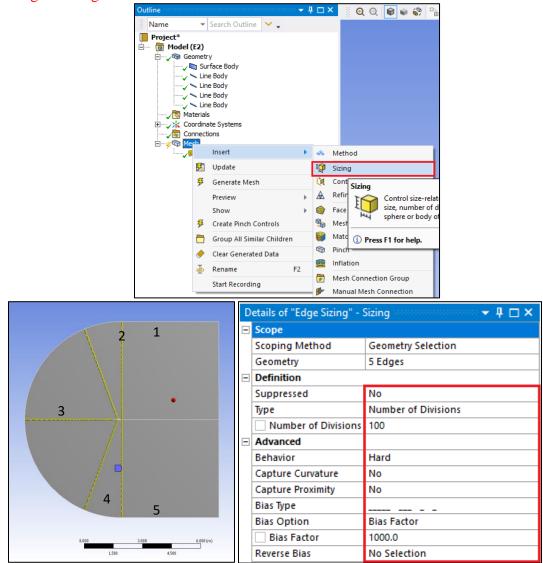
5.2. Right click on **Mesh** > **Insert** > **Face Meshing**. Select all six surfaces while holding Ctrl and click **Apply** in the yellow Geometry selection box.

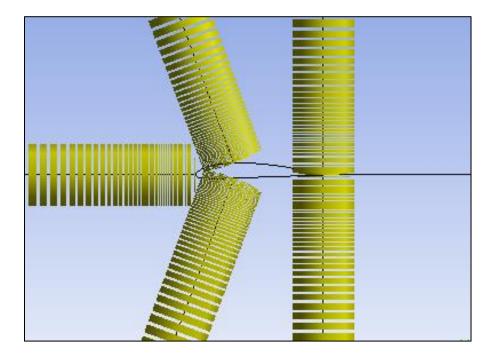
																			_			_
0	utline		्र म 🗆 ×	Q	Q	0	1	8	0.	\diamond	Q	0	Select	🔩 Mode	11 v	er 🖬 🖬 🖬		×yz 🕎	4	🛅 Clipboard 🕶	[Empty]	Ð
	Name	-	×																			
	Project*																					
E	🗄 🐻 Model (B																					
	E Geo	metr 1 Sur	/ face Body																			
		Line	Body																			
		Line	Body																			
			Body Body																			
	mat																					
	🗄 🗸 🦗 Coo	rdna	te Systems																			
	/ 🐨 Con		ons			•																
			Insert	-	_			Method				_										
		-				<u> </u>						-1										
		汉	Update				~	Sizing				- 1										
		۶	Generate Me	sh				Contact				- 1					71					
			Preview			> 2	€	Refinem	ent													
			Show			•		Face Me	shing													
		۶	Create Pinch	Controls		Q		Mesh Co		_												
			Group All Sim	ilar Chilo	dren			Match C	ontrol	Face												
		-	Clear General	ed Data		0	3	Pinch		Æ	b	nable 1	the genera e or mapp	tion								
		т. Т.	Rename		F2	. 5		Inflation		-			n selected									
		ЧD			12	- 1	-	Weld		—												
0	etails of "Mesh" Display	_	Start Recordi	ng		_ (8	Mesh Co	nnect	ا ()	Press F	1 for I	help.									
1	Display Display Style	10	se Geometry					Manual	Mesh (Conne	ction			-								
8	Defaults		se ocometry m				8	Mesh Ed	li t													
	Physics Preference	e N	lechanical					Mesh Nu		20		- 1				\checkmark						
	Element Order		rogram Cont									-										
	Element Size	D	efault (0.481					Contact			p	- 1										
5	Sizing Quality					0	-	Contact				_										
8	Inflation					0	_	Node M		roup												
8	Batch Connectio	ns				1	¥.	Node M	erge													
8	Advanced					•	•	Node M	ove													
3	Statistics											_										
									6	ч г .,	1	4	t	N.4		Face N ▼ 4		1				
								_		Га	cen	nesn	iing -	wappe	eu	race iv 👻 🛱						
							E	Sco	pe													
								Sco	ping	a Mi	etho	dG	Seomet	ry Sele	ctic	on						
								Geo	omet	trv		Г	A	oply		Cance	9					
							E	Def														
								Sup	pre	ssec	ł	N	lo									
								Maj	ope	d M	esh	Y	es									
								Met	hod	i		C	Juadril	aterals								
									nter	rnal	Nu.	D)efault									
								Cor	nstra	in E	Bour	N	lo									

5.3. Select the edge button.

Duplicate Copy Q Find	RNamed Selection @Images" C Coordinate System ☐Section Plane ☐Comment ☐Section Plane ☐Comment ☐Plane ☐Comment ☐Section Plane ☐Sectio	Full Green CRes Layo
Outline 👻 🖡	🗆 × 🔢 🝳 🔍 📦 😜 😘 🕞 🔾 × 🚸 🍳 🍳 🍭 🔍 Select 🤸 Mode 🖉 🏗 🛅 🛅 🛅 📾 📾 🍕 😤 👰 🧧	Clipboard •
Name Search Outline Project* Project* Project* Surface Body Vine Body	Face Meshing 2:33 PM Face Meshing Face Meshing Face Meshing Control to select multiply tedges on y model. Use the Chi Dutton or move button to select multiply Press F1 for help.	hold the

5.4. Right click on **Mesh > Insert > Sizing.** Select five lines below and click **Apply.** Change parameters as per below. NOTE: The divisions must be finer toward the airfoil. If they are not fine toward the airfoil you may need to change the bias direction by changing bias type for the edges having issue.

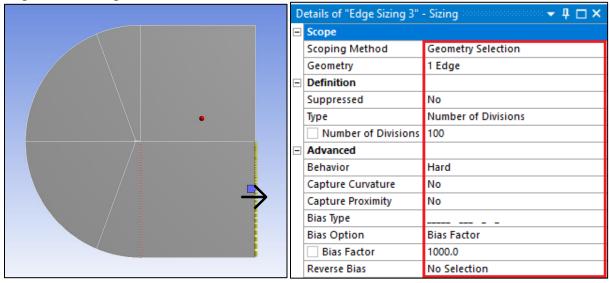




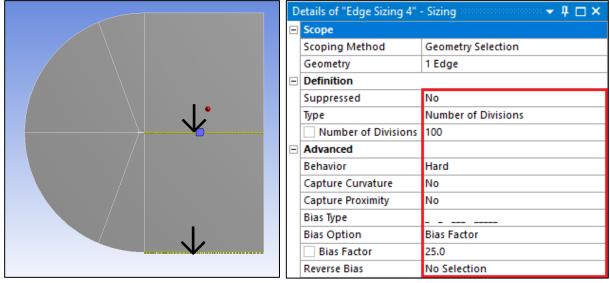
5.5. Right click on **Mesh > Insert > Sizing.** Select the line below and click **Apply.** Change parameters as per below. Divisons must be finer toward the airfoil.

		D	etails of "Edge Sizing 5" ·	- Sizing 👻 🖵 🗙			
		-	Scope				
			Scoping Method	Geometry Selection			
			Geometry	1 Edge			
	\rightarrow	-	Definition				
			Suppressed	No			
•			Туре	Number of Divisions			
			Number of Divisions	100			
		-	Advanced				
			Behavior	Hard			
			Capture Curvature	No			
			Capture Proximity	No			
			Bias Type				
			Bias Option	Bias Factor			
			Bias Factor	1000.0			
	_		Reverse Bias	No Selection			

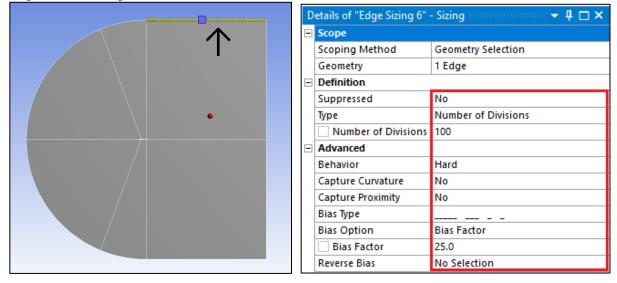
5.6. Right click on **Mesh > Insert > Sizing.** Select the line below and click **Apply.** Change parameters as per below. Divisons must be finer toward the airfoil.



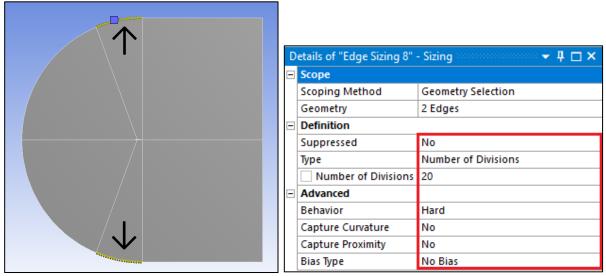
5.7. Right click on **Mesh > Insert > Sizing.** Select two lines below and click **Apply.** Change parameters as per below. Divisons must be finer toward the airfoil.



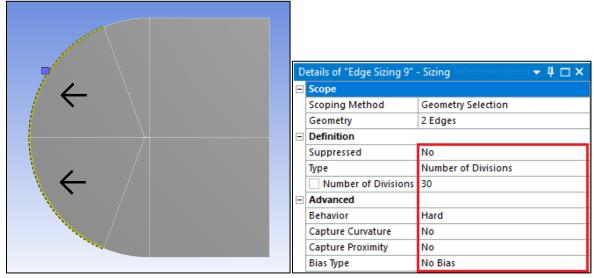
5.8. Right click on **Mesh > Insert > Sizing.** Select the line below and click **Apply.** Change parameters as per below. Divisons must be finer toward the airfoil.



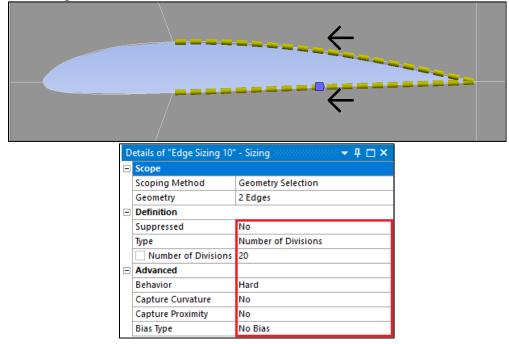
5.9. Right click on **Mesh > Insert > Sizing**. Select two lines below and click **Apply**. Change parameters as per below.



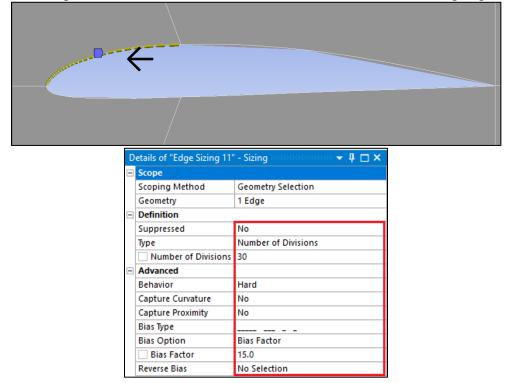
5.10. Right click on **Mesh > Insert > Sizing.** Select two lines below and click **Apply.** Change parameters as per below.



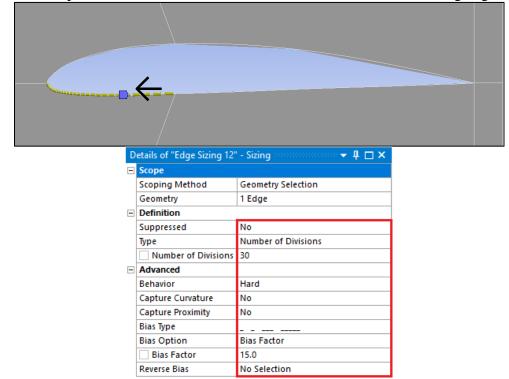
5.11. Right click on **Mesh > Insert > Sizing**. Select two lines below and click **Apply**. Change parameters as per below.



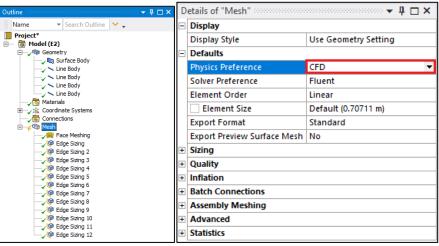
5.12. Right click on **Mesh > Insert > Sizing.** Selct the line below and click **Apply.** Change parameters as per below. Divisons must be finer toward the airfoil's leading edge.



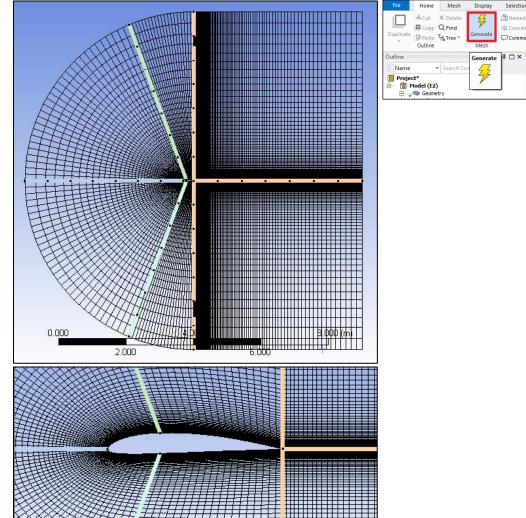
5.13. Right click on **Mesh > Insert > Sizing**. Selct the line below and click **Apply**. Change parameters as per below. Divisons must be finer toward the airfoil's leading edge.



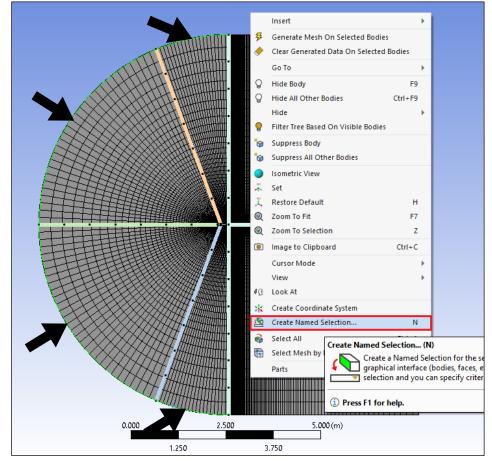
5.14. Click on **Mesh** under the **Outline** and under the **Details of "Mesh"** change the **Physics Preference** from **Mechanical** to **CFD**.



5.15. Click Generate Mesh. Your mesh should like the figures below.

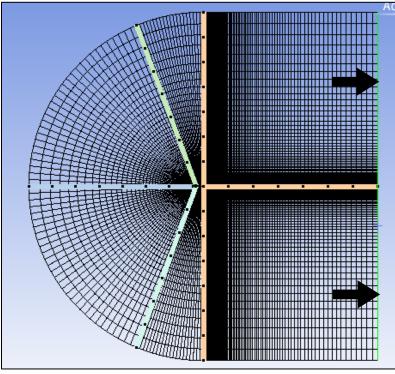


5.16. Select all the edge parts that make up the arc by holding down Ctrl and selecting them individually. Right click the selection and select **Create Named Selection**. Change the name to **inlet** and click OK.

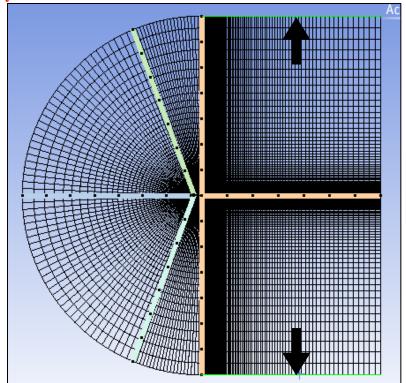


Selection Name	Х
inlet	×
 Apply selected geometry 	
 Apply geometry items of same: 	
Size	
🗌 Туре	
Location X	
Location Y	
Location Z	
Apply To Corresponding Mesh Nodes	
OK Cancel	1.0000444211

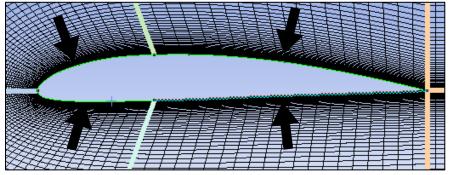
5.17. Select the vertical lines on the right side of the domain and right click it, then select **Create Named Selection**. Name this **outlet**.



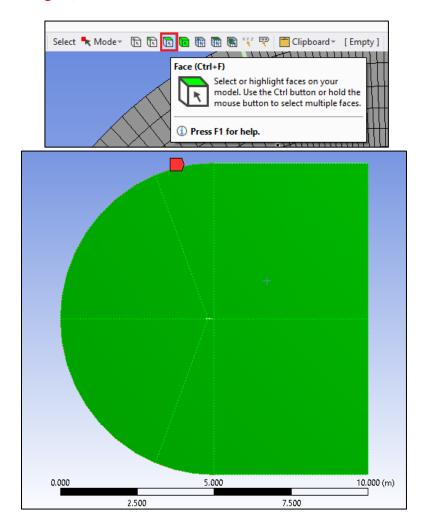
5.18. Select the two horizontal lines below, right click and **Create Named Selections**. Name them **symmetry**.



5.19. Select the four edges that make the airfoil, right click and **Create Named Selections**. Name them **airfoil**.



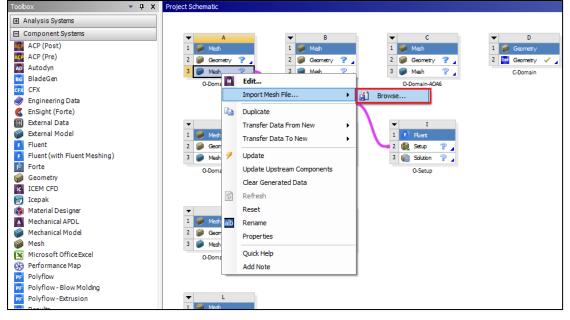
5.20. Select the six faces of the domain and right click them, select **Create Named Selections** (Make sure to select the **face selecting pointer** from the toolbox as shown at below to select "faces", not the "edges"). Name them **fluid**.



5.21. File > Save Project. Close Meshing window.

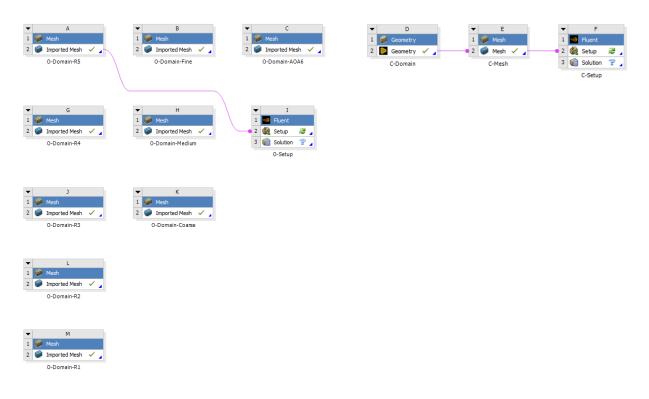
- D Е F 1 1 🧭 Geometry Mesh Fluent 1 2 2 Geometry \checkmark Mesh Set ¢° Edit... C-Mesh C-Domain D) Duplicate Transfer Data From New Transfer Data To New ۲ Update Update Upstream Components Clear Generated Data
- 5.22. Right click on Mesh and click Update.

- **5.23.** Go to class website (<u>http://www.engineering.uiowa.edu/~me_160/</u>) and download the mesh files. You will need to extract the files from their compressed format, simply right click on the .rar file and select **Extract Here**.
- 5.24. Right click on Meshes for O-Domains and select Import Mesh File > Browse. Select the Mesh file corresponding to mesh name and click Open. You need to change the import file type from CFX Mesh Files(*.gtm;*.cfx) to Fluent Files(*.cas;*.msh;*.cas.gz;*msh.gz).



ightarrow 🕆 🔂 > Thi	s PC > Desktop > CFD Lab 2 Meshe	:5		√ Ō	Search CFD Lab 2 Meshes
Irganize 🔻 New folde	r				
	Name	Date modified	Туре	Size	
Quick access	AOA6.msh	7/24/2013 12:57 PM	MSH File	899 K	(B
🗄 Documents 🖈	coarse.msh	7/24/2013 12:57 PM	MSH File	1,165 K	(B
🕂 Downloads 🛛 🖈	Domain-R1.msh	7/24/2013 12:57 PM	MSH File	899 K	(B
📰 Pictures 🛛 🖈	Domain-R2.msh	7/24/2013 12:57 PM	MSH File	899 K	(B
9. TA	Domain-R3.msh	7/24/2013 12:57 PM	MSH File	899 K	(B
E Desktop	Domain-R4.msh	7/24/2013 12:57 PM	MSH File	899 K	(B
HW3	Domain-R5.msh	7/24/2013 12:57 PM	MSH File	899 K	(B
💻 sungtpark (\\home.	📄 fine.msh	7/24/2013 12:57 PM	MSH File	5,977 K	(B
	📄 medium.msh	7/24/2013 12:57 PM	MSH File	2,642 K	(B
This PC					
🎐 Network					
File na	me:			~	FLUENT Files(*.cas;*.msh;*.cas.
					Open 🔻 Cancel

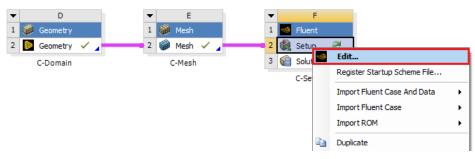
5.25. After completion of the previous step you will have a schematic as per below.



6. Setup

In this section you will create solver setups for C-mesh and O-Domain-R5. For the rest of the simulation, you will duplicate the setup from O-Domain-R5 case.

6.1. Right click Setup and select Edit.



6.2. Select **Double Precision** and click **Start**. After entering the Fluent, if you see any pop-up, you can ignore it by clicking **cancel**.

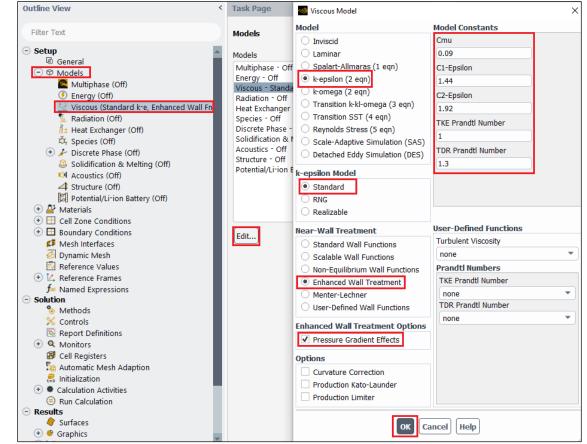
Fluent Launcher 2022 R1 (Setting Edit Only)	_		×
Fluent Launcher		<mark>/\</mark> n	sys
Simulate a wide range of steady and transient industrial a general-purpose setup, solve, and post-processing capabil including advanced physics models for multiphase, combus and more.	ities of A	NSYS F	luent
Dimension			
2D			
○ зр			
Options			
Double Precision	n		
0		ling	
Display Mesh Afl			
Do not show this	s panel a	again	
Load ACT			
Parallel (Local Mac	chine)		
Solver Processes		1	÷
Solver GPGPUs per M	Machine	0	\$
✓ Show More Options ✓ Show Learning Resources Start Cancel Help	;		
	•		

6.3. Tree > Setup > General > Check. Click Check and check the output (red box shown below) to see if there is any error for Mesh.

C-Mesh		~	
Outline View	Task Page	×	Mesh
Filter Text	General	(
○ Setup ● Models ● Doudary Conditions ○ Dynamic Mesh ■ Reference Values ● Kerence Frames ✓ Controls ● Solution ● Methods ✓ Controls ◎ Regort Definitions ● Calculation Activities ◎ Cell Registers ● Initialization ● Calculation Activities ● Graphics ● Joins ● Surfaces ● Secuptics ● Reports ● Reports ● Parameters & Customization	Display Ur Solver Type Pressure-Based Density-Based Time Steady Transient Gravity	heck Report Quality itts Velocity Formulation Absolute Relative 2D Space Planar Axisymmetric Axisymmetric Swirl	
			<pre>x-coordinate: min (m) = -4.959304e+00, max (m) = 5.304759e+00 y-coordinate: min (m) = -4.9593049e+00, max (m) = 5.000151e+00 Volume statistics: minimum volume (m3): 1.804453e-07 maximum volume (m3): 6.286124e-02 total volume (m3): 6.286124e-02 Total volume (m3): 6.286124e-01 Face area statistics: minimum face area (m2): 3.167363e-04 maximum face area (m2): 3.37279E-01 Checking mesh</pre>

O-Domain-R5 Mesh

Outline View	Task Page	8 🖸 Mesh
Filter Text	General	
Setup	Mesh Scale Check Report Quality Display Units Solver Solver Ype Velocity Formulation Pressure-Based Density-Based Relative	
	Time 2D Space © Steady © Flanar O Transient Axisymmetric Axisymmetric Swith Axisymmetric Swith	
Surfaces Surfaces Graphics Plots Plots Animations Reports Parameters & Customization	Gravity	
		Console x-coordinate: min (m) = -5.000000e+00, max (m) = 5.000000e+00 y-coordinate: min (m) = -5.00000e+00, max (m) = 5.000000e+00 Volume statistics: minimum volume (m3): 1.886282e-08 maximum volume (m3): 1.801741e-01 total volume (m3): 7.888658-01 Face area statistics: minimum face area (m2): 2.690604e-04 maximum face area (m2): 2.690604e-01 Checking mesh Done.



6.4. Tree > Setup > Models > Viscous > Edit... Choose the options below and click Ok

6.5. Tree > Setup > Materials > Fluid > air > Create/Edit. Change Density and Viscosity to experimental values and click Change/Create then click Close.

Outline View	Task Page	Create/Edit Materials				×
Filter Text	Materials	Name		Material Type		Order Materials by
- Setup		air		fluid	*	Name
General	Materials	Chemical Formula		Fluent Fluid Materials		Chemical Formula
📀 🍄 Models	Fluid			air	-	Fluent Database
Aterials	air			Mixture		
Eluid Fluid Air	Solid			none	*	User-Defined Database
(*) 🖉 Solid	aluminum		Properties			, ,
Cell Zone Conditions			Density (kg/m3)	▼ Edit.	
🔹 🖽 Boundary Conditions			Density (kg/m3	Constant	• Edit.	
🕗 Dynamic Mesh				1.2089		
🖹 Reference Values			Viscosity (kg/m-s) constant	▼ Edit.	
🛞 🛃 Reference Frames			viscosity (kg/m s			
Named Expressions				1.815e-05		
 Solution Methods 						
X Controls						
Report Definitions						
Q Monitors						
Cell Registers						
🛃 Initialization						
🕀 🗮 Calculation Activities			Chang	ge/Create Delete Close Help		
Run Calculation			minimum voia	me (ms). 1.001155e-07		
Results				me (m3): 6.286124e-02 me (m3): 8.925418e+01		
Graphics			Face area stat			
(+) L Plots				area (m2): 3.167363e-04		
Animations	Create/Edit Delet	e		area (m2): 3.372978e-01		
📀 🔜 Reports			Checking mesh. Done.			
Parameters & Customization						

6.6. Tree > Setup > Boundary Conditions > inlet > Edit... Change velocity to experimental condition and the rest of the parameters to values shown below and click OK(Apply).

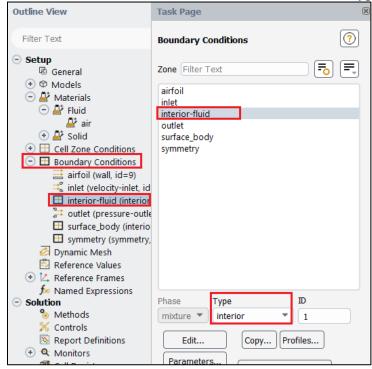
Outline View	Task Page	Velocity Inlet X				
Filter Text	Boundary Conditions	Zone Name inlet				
Setup Image: Constraint of the set of the		Momentum Thermal Radiation Species DPM Multiphase Potential UDS Velocity Specification Method Components * * Reference Frame Absolute * * * * Supersonic/Initial Gauge Pressure (pascal) 0 * </td				
 ⇒ surface_body (interio ⇒ symmetry (symmetry, ⊘ Dynamic Mesh ⇒ Reference Values ◆ L, Reference Frames ∱∞ Named Expressions 		Turbulent Kinetic Energy (m2/s2) 0.08 Turbulent Dissipation Rate (m2/s3) 7.4 OK Cancel Help				
Solution ⁶ Methods [×] Controls [∞] Report Definitions [•] Q. Monitors [®] Cell Registers [®] Initialization [•] Calculation Activities [©] Calculation Activities [●] Calculation	Display Mesh]	minimum volume (m3): 1.884453e-07				

Inlet Boundary Condition					
Variable $u (m/s)$ $v (m/s)$ $P (Pa)$ $k (m^2/s^2)$ $e(m^2/s^3)$					
Magnitude	7.04	0	0	0.08	7.4
Zero Gradient	-	-	-	-	-

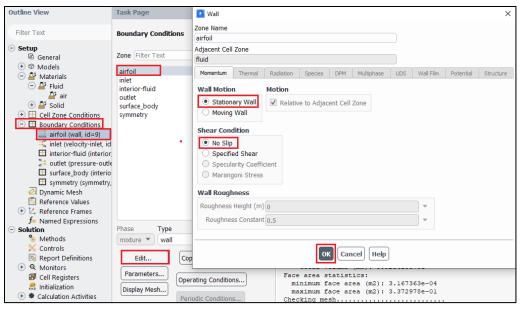
6.7. Tree > Setup > Boundary Conditions > outlet > Edit... Change Turbulence parameters to values shown below and click **OK(Apply)**.

Outline View	Task Page		Pressure	Pressure Outlet ×						×
Filter Text			Zone Name							
Filter Text	Boundary Conditions		outlet							
⊙ Setup	Zone Filter Text		Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
 Models Attended 	airfoil		1	Backflow Re	ference Fran	me Absolut	e			~
🕞 🔐 Fluid	inlet interior-fluid			Gauge Pr	essure (paso	cal) o				
🚔 air	outlet			Pressure I	Profile Multip	lier 1				-
🕒 🔐 Solid	surface_body					-				
Cell Zone Conditions Boundary Conditions	symmetry •		Backflow Dir	ection Speci	fication Meth	od Normal	to Boundary	1		
airfoil (wall, id=9)			Bac	flow Pressu	re Specificati	ion Total Pr	essure			-
inlet (velocity-inlet, id			Average	Pressure Sp	ecification					
interior-fluid (interior			Target N	Mass Flow Ri	to					
🗦 outlet (pressure-outl			Turbulenc							
surface_body (interio			Turbulenc							
symmetry (symmetry, Ø Dynamic Mesh				Specifi	cation Metho	d Intensity	and Length	Scale		•
Reference Values			Backfl	ow Turbulen	t Intensity (%	6) 3.25				-
⊕ ¼ Reference Frames			Backflow 1	urbulent Lei	ngth Scale (n	n) 0.0035				
fx Named Expressions			_		· ·					
 Solution 	Phase Type					_				
Methods	mixture 🔻 pres	sure-ou	tl		I	OK Can	cel Help			
X Controls	Edit	Com								
	Euit	Copy	Profiles	J				6.286124e- 8.925418e+		
Cell Registers	Parameters	0	No. Conditions			rea stati				
🛃 Initialization	Display Mesh	Opera	ting Conditions	·]): 3.16736		
💿 🏶 Calculation Activities	Dispidy Mesil	Perio	dic Conditions.): 3.37297		
Run Calculation					Done.	ig meen.				
		Ou+1	at Bour	dam	onditi	on				
		Outi	et Bour	iuai y C	Jonann	on				

- Variable u (m/s) v (m/s) P (Pa) Intensity (%) Length scale (m) Magnitude 3.25 0.0035 0 _ Zero Gradient Y Y ---
- **6.8.** Tree > Setup > Boundary Conditions > interior-fluid/surface body (you will have surface body in addition to default-interior for C-mesh). Make sure those Types are interior.

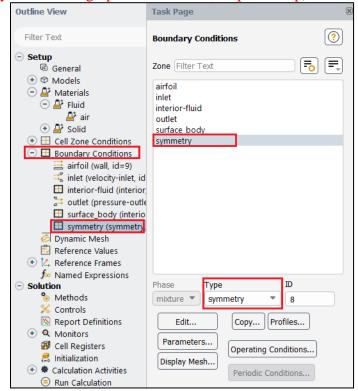


6.9. Tree > Setup > Boundary Conditions > airfoil. Make sure wall is selected under Type.



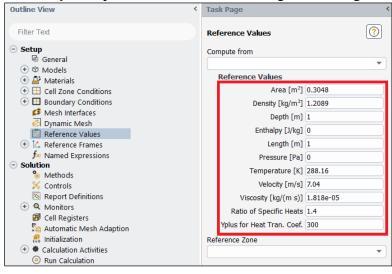
Airfoil Boundary Condition						
Variable $u (m/s)$ $v (m/s)$ $P (Pa)$ $k (m^2/s^2)$ $e (m^2/s^3)$						
Magnitude	0	0	-	0	0	
Zero Gradient Y						

6.10. Tree > Setup> Boundary Conditions > Symmetry. Make sure symmetry is selected under Type. (If you are setting up O-Domain-R5, skip this step)



Symmetry Boundary Condition						
Variable $u (m/s)$ $v (m/s)$ $P (Pa)$ $k (m^2/s^2)$ $e (m^2/s^3)$						
Magnitude	-	0	0	-	-	
Zero Gradient Y N - Y Y						

6.11. Tree > Setup > Reference Values. Change Reference Values of Density, Temperature, Velocity, and Viscosity to experimental values and change remaining values as per below.



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

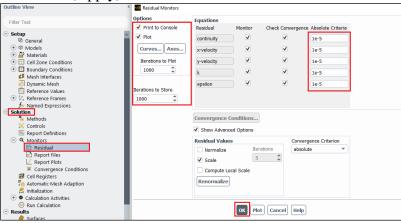


Outline View	< Task Page <
Filter Text	Solution Methods
- Setup	Pressure-Velocity Coupling
C General	Scheme
 	SIMPLE
Materials Arrow Conditions	Flux Type
	Rhie-Chow: distance based 🔹 🗌 Auto Select
🛤 Mesh Interfaces	Spatial Discretization
Dynamic Mesh	Gradient
 Reference Values X. Reference Frames 	Green-Gauss Cell Based
Named Expressions	Pressure
 Solution 	Standard
🐌 Methods	Momentum
🔀 Controls	Second Order Upwind
Report Definitions	Turbulent Kinetic Energy
 Monitors Cell Registers 	
Automatic Mesh Adaption	Second Order Upwind
	Turbulent Dissipation Rate
💿 🗭 Calculation Activities	Second Order Upwind
Run Calculation	Transient Formulation
Results	•
 Surfaces Graphics 	Non-Iterative Time Advancement
Graphics Plots	
Animations	Frozen Flux Formulation
💽 📑 Reports	Warped-Face Gradient Correction

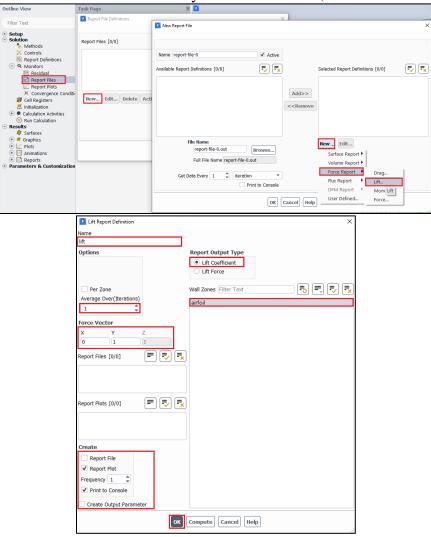
6.13. Tree > Solution > Controls. Change the Under-Relaxation Factors for, Momentum, Turbulent Kinetic Energy, and Turbulent Dissipation Rate to the values below. If your solution diverges try reducing under-relaxation factors.

Outline View	Task Page	×
Filter Text	Solution Controls	?
 Setup General Models A Materials A fuid air A Solid Cell Zone Conditions Boundary Conditions Dynamic Mesh Reference Values X. Reference Values X. Reference Frames Named Expressions Solution Methods Cell Registers Initialization Cell Registers Initialization Cell Registers Initialization Results Surfaces Y Graphics Y Intersection 	Under-Relaxation Factors Pressure 0.3 Density 1 Body Forces 1 Momentum 0.5 Turbulent Kinetic Energy 0.5 Turbulent Dissipation Rate 0.5 Turbulent Viscosity 1 Default Equations Limits Advance	ed

6.14. Tree > Solution > Monitors > Residual... Change convergence criterions for all 5 equations and click OK(Apply).



6.15. Tree > Solution > Monitors > Report files > New... Then, inside the New Report File, New > Force Report > Lift... Set parameters as per below, make sure to select airfoil and click OK. This will show the time history of lift coefficient. (This section is only for C mesh)



6.16. (6.15 continued) In **New Report File** dialog, specify the file location of the lift coefficient time-history file by clicking **Browse...** Check **Print to Console** and click **OK(Apply).** Close **Report File Definitions** dialog as well.

Name report-file-0				
Available Report Definitions [0/1]		Selected Report Definitions [0/0]	=	
lift	Add>> < <remove< td=""><td></td><td></td></remove<>			
File Name H:/9. TA/CFDLab2/axialve Browse		New , Edit		
Get Data Every 1 🗘 iteration 💌				
Cancel Help				

6.17. Tree > Solution > Initialization. Change the X Velocity to the experimental value and the rest of the parameters as per below and click Initialize.

Jutline View	Task Page		
Filter Text	Solution Initialization	?	
 Setup Solution Methods Controls Report Definitions A Monitors Cell Registers Initialization Calculation Activities Run Calculation Results Graphics Plots Plots Animations Reports Parameters & Customization 	Initialization Methods Hybrid Initialization Standard Initialization Compute from Reference Frame Relative to Cell Zone Absolute Initial Values Gauge Pressure (pascal) X Velocity (m/s) 7.04 Y Velocity (m/s) 0 Turbulent Kinetic Energy (m2/s2) 0.08 Turbulent Dissipation Rate (m2/s3) 7.4 Initialize Reset Patch		

6.18. Tree > Solution > Run Calculation. Change Number of Iterations to 10,000 and click Calculate. You will see the residual and lift coefficient figures for the C type mesh in windows 1 and 2, respectively. Save the lift coefficient figure for C mesh before continuing.

Outline View	Task Page
Filter Text	Run Calculation
Setup Solution Methods	Check Case Update Dynamic Mesh
Kethods Kontrols S Report Definitions	Options
• • Monitors	Data Sampling for Steady Statistics
Cell Registers	Sampling Interval
Calculation Activities Run Calculation	Iterations Sampled 0
 Results 	
 Surfaces Graphics 	Number of Iterations Reporting Interval
	10000
Scene	Profile Update Interval
 Animations Reports 	1
Parameters & Customization	Data File Quantities Acoustic Signals
	Calculate

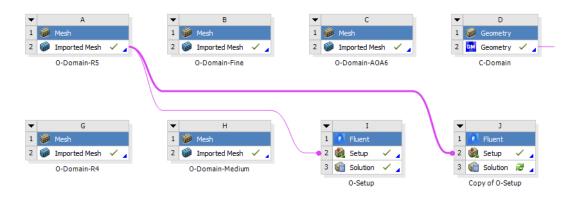
- **6.19.** File > Save Project. Close fluent. (Please complete setup for O-Domain-R5 and C-mesh before continuing).
- **6.20.** Once you completed the setups for Domain-R5 and C-mesh your schematic should look like the figure below. Next, we will copy O-setup and modify it where necessary for other cases.

A 1 Wesh 2 Propried Mesh V 0-Domain-R5	B 1 W Mesh 2 W Inported Mesh V C 0-Domain-Fine	C 1 Wesh 2 Wesh 2 Triported Mesh V 0-Domain-A0A6	▼ D 1 Secondary 2 Geometry ✓ ↓ C-Domain	E 1 Wesh 2 Mesh C-Mesh	Fluent 2 Setup 3 Solution C-Setup
	H Mesh Mesh @ Imported Mesh ✓ ↓ 0-Domain-Medium	I I Eucnt 2 Q Setup ✓ 3 Gesolution ✓ 0-Setup			
✓ 3 1 Weth 2 Weth 2 Deported Mesh ✓ 3 0-Domain-R3	 ✓ K 1 Mesh 2 Imported Mesh ✓ , O-Domain-Coarse 				
L 1 W Mech 2 Minported Mesh V 0-Domain-R2					
✓ M 1 ₩ Mesh 2 ♥ Imported Mesh ✓ ↓ 0-Domain-R1					

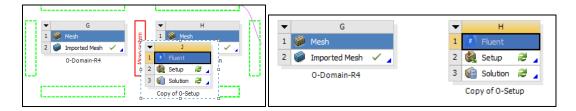
▼		А		-		В		•		С			•	D	
1	۲	Mesh		1	۲	Mesh		1	۲	Mesh			1 (🎾 Geometr	
2	۲	Imported Mesh	A	2	۲	Imported Mesh	✓ ₄	2	6	Imported M	lesh 🗸 🖌		2	Geometr	у 🖌 .
		O-Domain-R5	$ \setminus $			0-Domain-Fine			C	D-Domain-A	IOA6			C-Domai	n
• 1		G Mesh		▼ 1		H Mesh		[▼ 1	I Seluent	_				
2		Imported Mesh	× .	2		Imported Mesh	 _ 	_	2	Setup	× .				
		0-Domain-R4			0-	-Domain-Medium	1		3	0-	Edit Register Sta Import Flue Import Flue Import RON	nt Case An nt Case			
•		J		-		К	_				-		_	,	
1	۲	Mesh		1		Mesh				ú)	Duplicate				
2	۲	Imported Mesh	× .	2	۲	Imported Mesh	× .				Transfer Da	ata From Ne	w	•	
		O-Domain-R3			0)-Domain-Coarse					Transfer Da	ata To New		• • I	

6.21. Right click on O-setup and click **Duplicate** as per below. This will create a new setup.

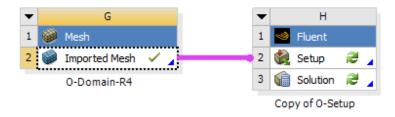
6.22. Select the connection between O-Domain-R5 and copy of O-setup then hit delete button on your keyboard.



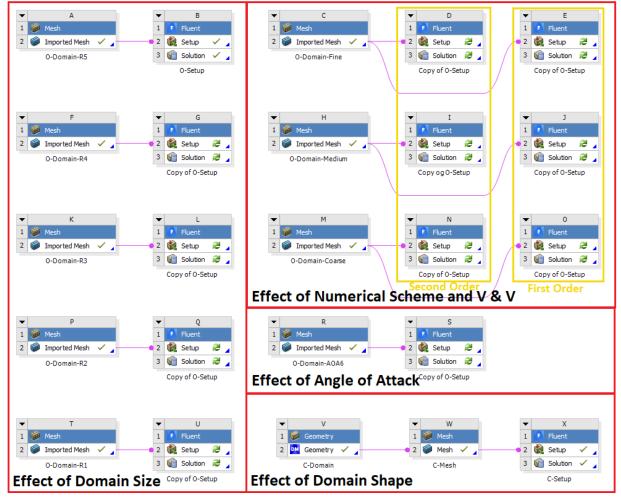
6.23. Drag and move the copy of O-setup next to Domain –R4 as per below: <u>click on the name</u> <u>of the box (Shaded with Blue)</u> when dragging components (e.g. click and drag Fluent).



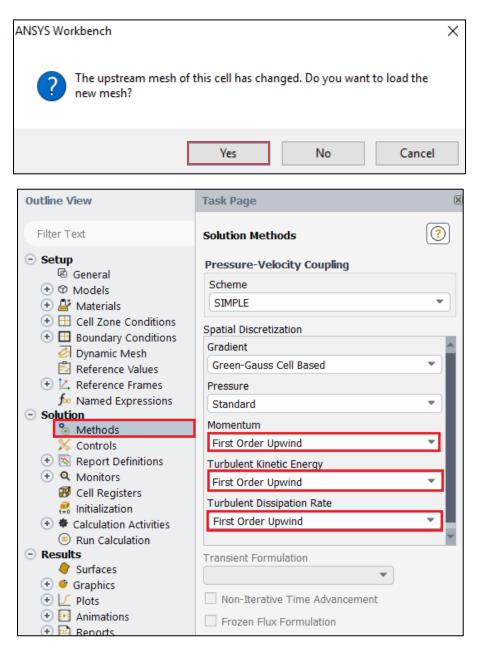
6.24. Create a new connection between copy of O-setup and Domain-R4 as per below. You have copied the setup from Domain-R5 to Domain-R4. Since they have the same setup, all you have to do is **initializing** and **running** inside the **Setup**.



6.25. Repeat the process for copying setup and get the project schematic as per below.



6.26. Setups for rest of the simulations are the same except for 3 simulations labeled with "First Order" in the figure above at V&V section. We need to change the scheme from second order to first order. Simply open fluent of cases with first order schemes. You will get message below, just click yes. Tree > Solution > Methods. Modify solution methods as per below and close Fluent.



6.27. Now you can run all the cases and save the ANSYS file. For this Lab2 only, find one partner in the class to form a group. One student will run V&V using first order upwind scheme, the other will use 2nd order upwind scheme. You will need to run rest of the cases by yourself.

7. Results

This section shows how to analyze your results. You do not need to do all the analysis for every case. Please read the exercises before continuing.

7.1. Saving Picture

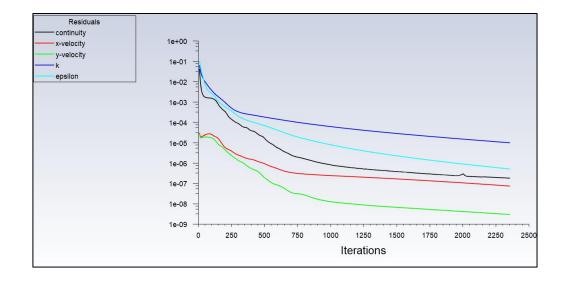
File > Save Picture. Choose options and click Save.

	<u>F</u> ile	Domain	
	Refresh	Input Data	
	Recorde	d Mesh Operation	IS
	Save Pro	ject	
	Reload		
	Sync Wo	rkbench	
	Read		•
	Write		•
	Import		•
	Export		•
	Solution	Files	
	Interpola	ate	
	EM Map	ping	•
	FSI Map	ping	•
	Save Pict		
	Data File	Quantiti Save Pic	ture
	Preferen	ces	
	Start Pag	ge	
	Close W	ithout Save	
	Close Flu	ient	
Format	Coloring	File Type	Resolution
O EPS	Color	Raster	Use Window Resolution
⊖ JPEG	◯ Gray Scale	O Vector	_
	 Monochrome 		Width 960 🌻
			Height 720 🌲
	Options		
	✓ Landscape Or	Windo	w Dump Command
	✓ White Backgro	impor	t -window %w
Window Dump	traite backgr		
	Save Apply	Preview Clo	se Help
	(1444)		()

7.2. Displaying Residuals

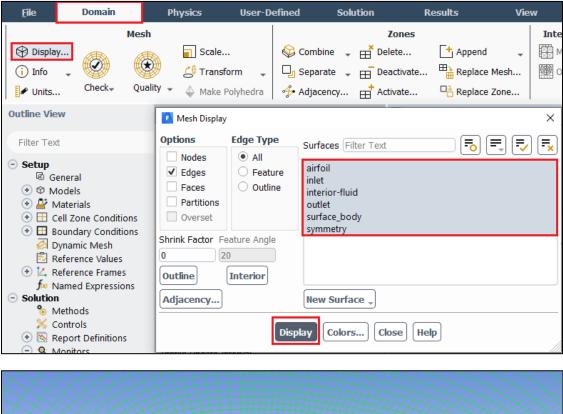
Tree > Solution > Monitors > Residuals > Plot. Save Picture with 7.1.

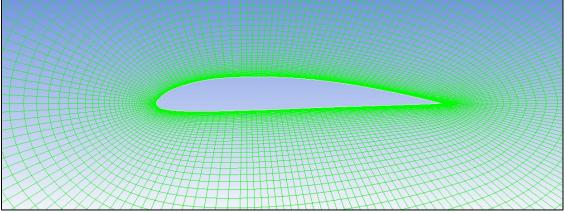
Options	Equations				
✓ Print to Console	Residual	Monitor	Check Conver	rgence Absolute Crite	ria
✓ Plot	continuity		✓	1e-05	
Window	x-velocity	v	✓	1e-05	
1 Curves Axes	y-velocity		\checkmark	1e-05	
Iterations to Plot	k	✓	✓	1e-05	
1000	epsilon	v	✓	1e-05	
Iterations to Store	Residual Values		Ci	onvergence Criterion	
	Normalize	Iterati	ons a	bsolute	•
	✓ Scale			Convergence Condit	ions
	Compute Local	Scale			
ок	Plot	e Cancel (Help		



7.3. Displaying Mesh

Setting Up Domain > Display. Save Picture with 7.1.





7.4. Printing Forces

Tree > Results > Reports > Forces. You can write the result as well by clicking **Write...** and save the data as a file at your working directory.

Options Forces Moments Center of Pressure Save Output Parameter	Direction Vector	Wall Zones Filter Text	
	Print Wri	te Close Help	

Drag (Direction Vector=[1, 0])

Lift (Direction Vector=[0, 1])

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

Example of Printed Drag (C-Mesh)

Forces Zone airfoil	Forces (n) Pressure (0.065176187 3.	2684099 0)		Viscous (0.14415489 0.005	59822478 0)		Total (0.20933108 3.2743921 0)	Coefficients Fressure (0.0071378644 0.35794464 0)
Net	(0.065176187 3.	2684099 0)		(0.14415489 0.005	59822478 0)		(0.20933108 3.2743921 0)	(0.0071378644 0.35794464 0)
Forces - Direction Vector Zone airfoil	(1 0 0) Forces (n) Pressure 0.065176187	Viscous 0.14415489	Total 0.20933108	Coefficients Pressure 0.0071378644	Viscous 0.015787332	Total 0.022925196		
Net	0.065176187	0.14415489	0.20933108	0.0071378644	0.015787332	0.022925196		

Example of Printed Lift (C-Mesh)

Forces Zone airfoil	Forces (n) Pressure (0.065176187 3.	2684099 0)		Viscous (0.14415489 0.005	59822478 0)		Total (0.20933108 3.2743921 0)	Coefficients Pressure (0.0071378644 0.35794464 0)
Net	(0.065176187 3.	2684099 0)		(0.14415489 0.005	59822478 0)		(0.20933108 3.2743921 0)	(0.0071378644 0.35794464 0)
Forces - Direction Vector Zone airfoil	(0 1 0) Forces (n) Pressure 3.2684099	Viscous 0.0059822478	Total 3.2743921	Coefficients Pressure 0.35794464	Viscous 0.00065515452	Total 0.35859979		
Net	3.2684099	0.0059822478	3.2743921	0.35794464	0.00065515452	0.35859979		

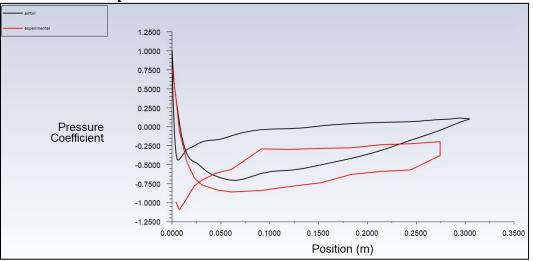
7.5. Plotting Pressure Coefficient on Airfoil Surface

Tree > Results > Plots > XY Plot. Select	parameter as per below and click Plot.
--	--

XY Plot Name				
xy-plot-1				
Options		Plot Direction	Y Axis Function	
✓ Node Values		XI	Pressure	-
Position on X Axis		Y O	Pressure Coefficient	-
Position on Y Axis		ZO	X Axis Function	
Order Points			Direction Vector	•
File Data [0/0]		Load File Free Data	Surfaces Filter Text	5 F
	Save/Plot	Axes Curv	es) Close Help	

To plot experimental data on top of CFD result, click Load File... and load the experimental pressure coefficient file, then click Plot.

XY Plot Name		
xy-plot-1		
Options	Plot Direction	Y Axis Function
✓ Node Values	X [1	Pressure 💌
Position on X Axis	YO	Pressure Coefficient
Position on Y Axis	Z [0]	X Axis Function
Order Points		Direction Vector
File Data [1/1] = = =	Load File Free Data	Surfaces Filter Text Fo F F F
		symmetry New Surface
Save/Plot	Axes Curve	s) Close Help



Example of Plot for Pressure Coefficient on Airfoil Surface

7.6. Plotting Pressure Contour

Tree > Results > Graphics > Contours. Check Filled, select Static Pressure and click Display.

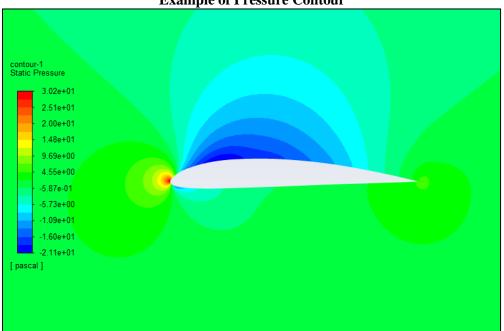
Contour Name contour-1	
Options	Contours of
 Filled Node Values Contour Lines Global Range Auto Range Clip to Range Draw Profiles Draw Mesh 	Pressure Static Pressure Min (pascal) Max (pascal) -21.14123 30.24535 Surfaces Filter Text airfoil fluid
Coloring Banded Smooth Colormap Options	inlet interior-fluid outlet surface_body symmetry
Sa	ve/Display Compute Close Help

Among **Surfaces**, the surface **fluid** can be added in the list by allocating zone name for the existing fluid zone (**fluid** isn't on the list by default). **Setting Up Domain > Surface > Create > Zone**.

<u>F</u> ile Don	nain	Physics	User-Defined	Solution	Results	Vie	w F	Parallel Desig	n 🔺	
Display Display Display Display Che		Scale	m 🗣 🖵	Zone Combine → 📅 Delete Separate → 📅 Deactiva Adjacency 🛱 Activate	te 📇 Append		Interfaces Mesh Werset	Mesh Models Dynamic Mesh Mixing Planes Turbo Topology	Adapt Refine / Coarsen	Surface + Create Zone Partition Zone
Outline View		Task Page				χ	1 Lt	711114	Conto	Point e (p

You can add **fluid** by highlighting **fluid** from the zone list and clicking **Create**. You can now go back to contour part and finish plotting.

Zone Filter Text 🔂 🖶 🗮	Surfaces Filter Text
airfoil fluid inlet interior-fluid outlet surface_body symmetry	airfoil fluid inlet interior-fluid outlet surface_body symmetry
Create Manage	New Surface Name fluid



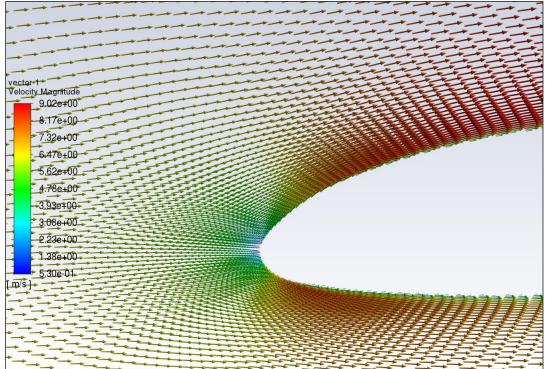
Example of Pressure Contour

7.7. Plotting Velocity Vectors

Vector Name vector-1	
Options	Vectors of
✓ Global Range	Velocity -
Auto Range	Color by
Clip to Range	Velocity
 Auto Scale Draw Mesh 	Velocity Magnitude
Style	Min [m/s] Max [m/s]
3d arrow	0.5301474 9.021389
Scale Skip	Surfaces Filter Text 🔂 🗮 🗮
0.1 0 Vector Options Custom Vectors Colormap Options	airfoil inlet interior-fluid outlet surface_body symmetry
	Display State None Use Active New Surface -
Sa	ave/Display Compute Close Help

Tree > Results > Graphics > Vectors > Set Up. Click Display.

Example of Velocity Vector

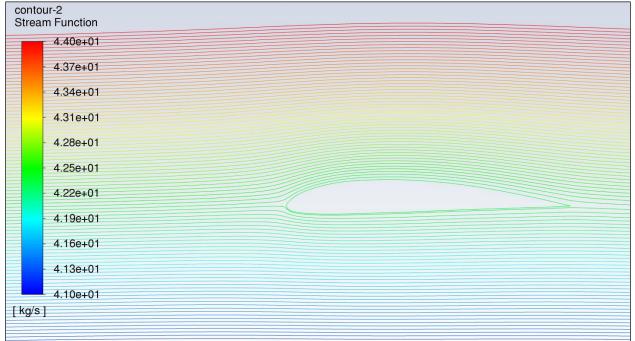


7.8. Plotting Streamline

Tree > Results > Graphics > Contours > Set Up... Select parameters as per below and click Display. You can adjust the Min and Max to get a better figure.

Contour Name	
contour-2	
Options	Contours of
☐ Filled ✓ Node Values	Velocity Stream Function
Contour Lines Global Range Auto Range Clip to Range	Min [kg/s] Max [kg/s] 41 44 Surfaces Filter Text = 0 = 0 = 0 = 0 = 0 = 0
Draw Profiles	airfoil fluid inlet
Coloring Banded Smooth 	interior-fluid outlet surface_body symmetry
Colormap Options	Display State None Use Active New Surface
Sa	ve/Display Compute Close Help

Example of Streamline



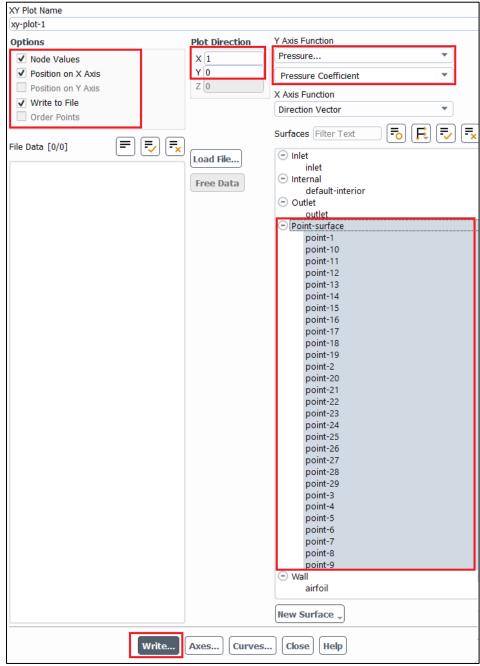
8. Verification and Validation (For V&V Simulations Only)

8.1. From the **Project Schematic**, right click on the **Fluent Solution** and select **Edit...** from the dropdown menu

Please make 29 Points manually using below Points.

File Domain Physics User-Defined Solution Image: Solution Mask Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution Image: Solution <t< th=""><th>ctivate</th><th>sen Surface + Create Partition Partition Partition Point Line/ Point Plane Guadric So-Cip Transform Point Create Close Help</th></t<>	ctivate	sen Surface + Create Partition Partition Partition Point Line/ Point Plane Guadric So-Cip Transform Point Create Close Help
	Х	Y
Point 1	0.000000	0.000000
Point 2	0.003810	0.006121
Point 3	0.007620	0.009357
Point 4	0.015200	0.013944
Point 5	0.022860	0.017059
Point 6	0.030500	0.019640
Point 7	0.045720	0.023500
Point 8	0.061000	0.026059
Point 9	0.091440	0.028093
Point 10	0.122000	0.028242
Point 11	0.152400	0.026656
Point 12	0.183000	0.023618
Point 13	0.213360	0.019255
Point 14	0.244000	0.013626
Point 15	0.274320	0.007302
Point 16	0.274320	-0.001214
Point 17	0.244000	-0.002221
Point 18	0.213360	-0.003142
Point 19	0.183000	-0.004193
Point 20	0.152400	-0.005436
Point 21	0.122000	-0.006680
Point 22	0.091440	-0.007523
Point 23	0.061000	-0.008457
Point 24	0.045720	-0.008685
Point 25	0.030500	-0.008307
Point 26	0.022860	-0.007929
Point 27	0.015200	-0.007243
Point 28	0.007620	-0.005942
Point 29	0.003810	-0.004674

8.2. Tree > Results > Plots > XY Plot > Setup... Select parameters as per below, make sure to select points 1 through 29, and click Write. Name the file for future reference. (This only needs to be done for coarse, medium, and fine manual meshes, which are used for V&V calculations. It is not needed for other mesh.)



8.3. Open the V&V Excel.

8.4. Copy and paste the pressure coefficients into the proper sheet corresponding to the mesh size. To do this open the saved pressure coefficient data in TextPad, use the "**Ctrl** + **A**" function to select all, then right click and select **Copy**.

TextPad - C:\Users\mconger\Desktop\Medium Grid Press Coeff (29 pts)		
File Edit Search View Tools Macros Configure Window Help		
i D 😅 🖬 🖶 🗗 🗶 🗐 🙏 🖻 🕲 으 오 🚍 🎫 😂 ୩ 🏈 ザ 斜 💇 💠	🙀 🔹 🗤 🕨 🔤 Find incrementally 🖟 🛈 🗌 Match case 🔤	
Document Selector # × Medium Grid Press Coeff (29 pts) ×		→ ×
Medum Gid Press Coef (23 pts) (Litle "Pressure Coefficient") (Labels "Position" "Pressure Coefficient") (Ky/key/label "point-1") 0 0.952383	at")	
)	Properties	=
((xy/key/label "point-10")	Cut	
0.12226 -0.562213	Сору	
	Paste	
((xy/key/label "point-11") 0.15282 -0.487262) ((xy/key/label "point-12") 0.18382 -0.405737	Cut Other Copy Other Inset Delet	
) ((xy/key/label "point-13") 0.21399 -0.300439)	Change Case → Transpose → Align →	
((xy/key/label "point-14")	Reformat	
0.24451 -0.165764	Block Select Mode Fill Block	
((xy/key/label "point-15")	Spelling	-
🗟 Explo 🖅 Docu 🧭 Clip L 🔄 🕯	Toggle Bookmark	► a
Search Results		후 ×
Search Results P Tool Output Copy the selection to the Clipboard		119 1 Read Ovr Block Sync Rec Caps

8.5. Paste this data into cell **A1** of the corresponding pressure coefficient tab. Right click on cell **A1** and select **Keep Text Only**. The cells to the right should auto populate extracting the correct data from the pasted data. If all the **x coordinate** cells are not green, there was an error in the pasting of the copied data.

	🚽 🤊 -	(°' • ∓			_	_	_	Template	V&V for a	irfoil with	data extractio	n.xlsx - Mic	rosoft	t Excel	_	_	_	_	_				x
F	ile H	ome	Insert Pa	ge Layout 🛛 Fo	rmulas	Dat	a Revi	ew View	Custo	m Comma	nds										۵ 🕜	- 6	p X
ľ	<u>لا</u> ا			11 × A A				🖶 Wrap Ter								•		Σ Au	itoSum +	A Z	A		
Pa	ste 🦼	BI	<u>U</u> - 🔛	• 👌 • <u>A</u> •			使使	📑 Merge &	Center *	\$ - %	· ·	Condit	ional	Format Cell as Table + Styles +	Inser	t Delete I	Format	2 CI		Sort & Filter ▼			
Clip	board G			G.			Alignme	nt	G	N	umber			Styles		Cells		Q2 4.1	Edi		Select *		
_			+ (e)	fx																-			~
	1		A			в	С	D	F	F	G	1		1		к			м	N		0	
1	(title "Pr		Coefficient")					-		9					ix.							-
				, Coefficient")						V&V	Data (Copy a	nd paste v	valu	e into V&V tem	olate)	1							
3														Pressure Coeff									
4	((xy/key)	/label "r	poin Paste 0	ptions:						1			0000		0								
5			Â			952393				2		0.0	0609		0								
6)									3			0939		0								- =
7	<i>`</i>		Keep Te	xt Only (T)						4		0 0.0	1401		0								
8	((xy/key)	/label "	point-10")							5		0.0	1712		0								
9				0.12226	-0.	562213				6		0.0	1969		0								
10)									7		0.0	2356		0								
11										8		0 0.0	2613		0								
12	((xy/key)	/label "p	point-11")							9		0 0.0	2818		0								
13				0.15282	-0.4	487262				10		0.0	2833		0								
14)									11		0.0	2674		0								
15										12		0.0	2371		0								
16	((xy/key)	/label "p	point-12")							13		0.0	1932		0								
17				0.18338	-0.4	405737				14		0 0.0	1371		0								
18)									15		0 0.0	0735		0								
19										16		0.0-	0122		0								
20	((xy/key)	/label "p	point-13")							17		0.0-	0223		0								
21				0.21399	-0.	300439				18		0.0-	0315		0								
22)									19		0.0-	0421		0								
23										20		0 -0.0	0545		0								
24	((xy/key)	/label "p	point-14")							21		0.0-	0670		0								
25				0.24451	-0.	165764				22		0.0-	0754	-	0								
26)									23			0848		0								
27										24		-	0871		0								
	((xy/key)	/label "p	point-15")							25		-	0833		0								
29				0.27504	-0.0	029141				26			0795		0								
30)									27			0727		0								
31										28			0596		0								
		/label "p	point-16")							29		-0.0	0467	r	0								
33				0.27504	0.0	845124																	
34															-								
		Equation	ns 🖉 V&V V	elocity / Verific	cation L	ift Coef	Fine	Grid Press Co	oeff Me	edium Gri	id Press Coe	ff Coars	e Grid	Press Coeff									
Rei	ady																	삐띠빌] 100%	Θ—		(÷.,

- **8.6.** Repeat these steps for the remaining mesh sizes.
- **8.7.** Once all the data is pasted into the three Press Coeff tabs, the V&V Velocity tab auto populates and calculates V&V values.
- **8.8.** Open the Verification Lift Coef tab and input values from y force report into the cells corresponding to the mesh size. The V&V values auto calculate.

9. Data Analysis and Discussion

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

9.1. Effect of domain size (+9)

Run 5 simulations using five different domain sizes using mesh O-Domain-R5, O-Domain-R4, O-Domain-R3, O-Domain-R2 and O-Domain-R1. Fill the table below with lift coefficient with their relative difference between two successive meshes. If the relative change between two successive domain sizes should be less than 1%, then which domain sizes will be enough large to make the CFD simulation results to be independent of the domain size?

Circle radius (m)	1	,	2		3		4		5
Lift Coefficient									
Relative change	N/A	()%	()%	()%	()%

• Figures need to be reported: None.

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

9.2. Effect of numerical scheme on Verification study for lift coefficient and validation of pressure coefficient (+17)

Use fine, medium and course meshes to conduct your V&V analysis. For this exercise only, find one partner in the class to form a group, one student will run V&V using first order upwind scheme, the other will use 2nd order upwind scheme. Then, you must borrow the figures/data from the other student and present in your lab report.

Based on verification results for lift coefficient, which numerical scheme is closer to the asymptotic range? Which numerical scheme has a lower mesh uncertainty? Discuss the verification and validation for pressure coefficient. For which locations of 29 points the pressure coefficient has been validated? For which locations the pressure coefficient has not been validated?

- Figures need to be reported: Figures in V&V excel sheet for 1st and 2nd order numerical schemes.
- Data need to be reported: Tables in V&V excel sheet for 1st and 2nd order numerical schemes.

9.3. C mesh generation (+5)

Follow the instructions in the manual and create the geometry and mesh manually. Does the lift coefficient for C mesh differ from O mesh? For iterative history of lift coefficient, what is the minimum iteration number for you to determine the lift coefficient has converged to a "constant" value?

- Figures need to be reported: C mesh generated by yourself, time history of lift coefficient.
- Data need to be reported: converged lift coefficient.

9.4. Effect of angle of attack on airfoil flow (+12)

Compare results from AOA6 (6 degree angle of attack) and O-Domain-R5 (0 degree angle of attack) meshes. O-Domain-R5 and AOA has the same mesh with the different angle of attack. Analyze the difference of the flow field. Which case has a higher lift coefficient, which has a higher drag coefficient?

• Figures need to be reported (for both attack angles): pressure contours, comparisons with EFD on pressure coefficient distribution, velocity vectors near airfoil surface, streamlines near the airfoil surface.

• Data need to be reported (for both attack angles): lift and drag coefficients.

9.5. Questions need to be answered when writing CFD report

- 9.5.1. Answer all the questions in exercises 1 to 4
- 9.5.2. Analyze the difference between CFD/EFD and possible error sources (+2)

10. 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 ← Section 9 (Page# 55) for CFD Lab	2	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