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

58:160 Intermediate Mechanics of Fluids CFD LAB 2

By Timur K. Dogan, Michael Conger, Maysam Mousaviraad, and Fred Stern IIHR-Hydroscience & Engineering The University of Iowa C. Maxwell Stanley Hydraulics Laboratory Iowa City, IA 52242-1585

1. Purpose

The Purpose of CFD Lab 2 is to simulate turbulent airfoil flows following "CFD process" by an interactive step-by-step approach and conduct verifications ANSYS software. Students will have "hands-on" experiences using ANSYS to conduct verification and validation for lift coefficient and pressure coefficient distributions, including effect of numerical scheme. Students will manually generate the "O" type and "C" type meshes 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 source of errors, and present results in the CFD Lab report.



Flow Chart for ANSYS

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 meshes. (Note: the figures are not in the exact scale as the true size of the domain and airfoil).

Parameter	Symbol	Unit	O-type	C-Type
Chord Length	С	m	0.3048	0.3048
Downstream length	Lo	m	-	12
Radius	Rc	m	5,4,3,2,1	5
Angle of attack	α	degree	0,6	0

Fable	1	-	Main	particulars
-------	---	---	------	-------------



In CFD Lab2, Boundary conditions for "C" type of meshes 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".

Grid	Domain	Radius [m]	Angle of Attack [degree]		
С	C-type				
O-fine-R5		5			
O-medium-R5		5			
O-course-R5			0		
O-course-R4		4	0		
O-course-R3	0-type	3			
O-course-R2	1	2			
O-course-R1		1			
O-course-R5-AOA6		5	6		

Table 2	- Grids
---------	---------

Table 3 - Simulation Matrix

Study	Grid
Domain size	O-course-R5, O-course-R4, O-course-R3, O-course-R2, O-course-R1
Numerical scheme on V&V	O-fine-R5, O-medium-R5
Domain shape	С
Angle of attack	O-course-R5-AOA6

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

	Grid	Flow	Convergence Limit	Figue	Data		
1	type=O-Domain-Study-coarse-R=5-aoa=0	Turbulent	1.00E-05	*	None		
2	type=O-Domain-Study-coarse-R=4-aoa=0	type=O-Domain-Study-coarse-R=4-aoa=0 Turbulent 1.00E-		None	None		
3	type=O-Domain-Study-coarse-R=3-aoa=0	Turbulent	1.00E-05	None	None		
4	type=O-Domain-Study-coarse-R=2-aoa=0	Turbulent	1.00E-05	None	None		
5	type=O-Domain-Study-coarse-R=1-aoa=0	Turbulent	1.00E-05	None	None		
6	type=O-V&V-R=5-aoa=0-fine	Turbulent	1.00E-05	Mesh,V&V Figures, Residuals	Pressure Coefficient (29 pts)		
7	type=O-V&V-R=5-aoa=0-medium	Turbulent	1.00E-05	Mesh,V&V Figures	Pressure Coefficient (29 pts)		
8	type=O-V&V-R=5-aoa=0-coarse	Turbulent	1.00E-05	Mesh,V&V Figures	Pressure Coefficient (29 pts)		
9	type=C-R=5-aoa=0	Turbulent	1.00E-05	Mesh	Lift Coefficient		
10	type=O-AOA-Study-coarse-R=5-aoa=6	Turbulent	1.00E-05	*	Lift and Drag Coefficient		
*	 Pressure Contour, comparison with EFD for pressure coefficient distribution, velocity vectors near airfoil surface, streamlines near airfoil surface 						

Table 4 - Figures and data sets needed to be saved

3. Open ANSYS Workbench and Layout Setup

3.1. Start > All Programs > ANSYS 14.5 > Workbench 14.5



3.2. Toolbox > Component Systems. Drag and drop Mesh and Fluent components to Project Schematic and connect the Mesh to the setup as per below.

Λ.	Insaved Project - Workbend	23.2	Works	wards.				- 0	x
FI	e View Tools Units	Extensions H	ielo						
1	New 🗃 Open 🛃 Save	Save As	d Import	e Reconnect	Refresh Project	🧭 Update Project	@Project	Compact Mode	
Tool	VAV	× 11 Y	Project Schema	No.		, ,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,	0		а х
E	Component Systems		Troject benenia						T ^
5	Autodyn								
l ă	CEX		-	۵		B			
l 🎽	Engineering Data		1	March	1	Eluant			
1 de	External Connection		2 🐼	Country 3		A fature and			
0	External Data		2 🖤	Geometry P		pg setup ₽,	4		
0	Finite ElementModeler		3 🐲	Mesh 🍟	- 30	Solution 🏆	4		
	Fluent			Mesh		Fluent			
	Fluent (with TGrid meshing)								
9	Geometry Fluent								
2	ICEM CFD								
	Mechanical APDL								
	Mechanical Model	E							
1	Microsoft Office Excel								
	Polyflow								
30	Polyflow - Blow Molding								
	Polyflow - Extrusion								
0	Results	-							
7	View Al /	Customize							
8	Ready						Show Progress	Show 5 Messag)es].::

3.3. Right click small down arrow in upper left corner of mesh component and select **Rename**. Rename the component according the grid and domain size.

æ	Refresh	1	•	Fluent		ľ
7	Update	2	¢,	Setup	~	
Ra I	Duplicate	3	C	Solution	~	
×	Clear Generated Data Delete Recreate Deleted Components			Fluent		
đ)	Rename Properties					

4. Mesh Importing

4.1. Right click on Mesh and select Import Mesh File....

Unsaved Project - Workbench					
File View Tools Units Ex	tensions Help				
🎦 New 📸 Open 🛃 Save 📓	Save As 👔 Import 🗟	9 Reconnect	<i>i</i> Refresh Project 🛛 🦩 Upda	i te Project 🔇 Project 🍯	Compact Mode
foolbox	▼ ₽ X Project Schematic				- q
Component Systems	A				
🐠 Autodyn					
CFX	•	A	▼ 6	3	
Engineering Data	1 鄃 M	lesh	1 💽 Fluen	t	
External Connection	2 🖗 0	oomotru 🦷	2 Sotur		
External Data		conico y a		· · · ·	
Finite Element Modeler	3 💓 M	esh 🚬	3 NG Soluti	on 😨 🖌	
E Fluent		Mesh 💾	Call	t.	
Fluent (with TGrid meshing)			Import Mesh File		
🥪 Geometry		Co.	Dupicate		
ICEM CFD		-	Transfer Data Ta Nam		
Mechanical APDL			Transfer Data to New •		
🍘 Mechanical Model		7	Update		
🍘 Mesh	=		Clear Generated Data		
Microsoft Office Excel			Pafrach		
29 Polyflow		10			
Polyflow - Blow Molding			Reset		
Polyflow - Extrusion		55	Rename		
Results	*		Properties		
Yiew All / Cu	stomize		Quick Help		
Double-click component to edit.			Add Note	Show Progress	Show 5 Messages

4.2. Select the O-automatic-course-R5-aoa-0 grid and click **Open**.

Draznize = New felder				8== -	E	6
organize • New folder				8== *		. 4
Favorites	Name	Date modified	Туре	Size		
🧮 Desktop	type=C-manual-R=5-aoa.msh	7/24/2013 12:57 PM	MSH File	611 KB		
〕 Downloads	type=O-automatic course-R=1-aoa=0.m	7/24/2013 12:57 PM	MSH File	899 KB		
Recent Places	type=0-automatic course-R=2-aoa=0.m	7/24/2013 12:57 PM	MSH File	899 KB		
	type=0-automatic course-R=3-aoa=0.m	7/24/2013 12:57 PM	MSH File	899 KB		
🔰 Libraries	type=O-automatic course-R=4-aoa=0.m	7/24/2013 12:57 PM	MSH File	899 KB		
Documents	type=O-automatic course-R=5-aoa=0.m	7/24/2013 12:57 PM	MSH File	899 KB		
J Music	type=O-automatic course-R=5-aoa=6.m	7/24/2013 12:57 PM	MSH File	899 KB		
Pictures	type=O-manual-R=5-aoa=0-fine.msh	7/24/2013 12:57 PM	MSH File	5,977 KB		
😸 Videos	type=O-manual-R=5-aoa=0-medium.msh	7/24/2013 12:57 PM	MSH File	2,642 KB		
	ype=O-manual-R=5-aoa=0-course.msh	7/24/2013 12:57 PM	MSH File	1,165 KB		
Computer IIHR_Image (C:)						
🖣 Network						
File na	me: type=O-automatic course-R=5-aoa=0.msh		•	FLUENT Files(*.cas;*.msl	n;*.cas	ų.

4.3. The layout file should look as follows, if the checkmark is a lightning bolt, try right clicking imported mesh and select **Update**.

۸	Unsaved Project - Workbench							x
F	File View Tools Units Extensions H	elp						
	🚹 New 📸 Open 🛃 Save 🔣 Save As	import av Reconnect	are Refresh Project	🗲 Upd	date Project	Project	Compact Mode	
То	obox 👻 🗘 🗙	Project Schematic					¥	ņх
E	Analysis Systems							
	Design Assessment							
6	Electric	▼ A		•	В			
	Explicit Dynamics	1 🥔 Mesh		1	Fluent			
B	Fluid Flow - Blow Molding (Polyflow)	2 M Imported Mesk	× -	2 🚵	Setun i			
	Fluid Flow-Extrusion(Polyflow)				Colution	-		
6	Fluid Flow (CFX)	Mesh		2 6	J Solution	8 🔺		
E	Fluid Flow (Fluent)				Fluent			
K	Fluid Flow (Polyflow)							
6	Harmonic Response							
2	1C Engine							
6	Linear Buckling							
0	Magnetostatic							
	🗑 Modal							
6	Random Vibration							
6	😗 Response Spectrum							
	Rigid Dynamics							
E	Static Structural							
	Steady-State Thermal							
7	View All / Customize							
	Ready					how Progress	Show 5 Messa	ges 🔡

5. Setup

5.1. Right click **Setup** and select **Edit...**



5.2. Select **Double Precision** and click **Ok**.



5.3. Click Check and check the output (red box shown below) for any errors.

B:Fluent Fluent [2d, d	o, pbns, lam] [ANSYS Academic Teaching Advanced]	
File Mesh Define Sc	lve Adapt Surface Display Report Parallel Vie	ew Help
i 📖 i 💕 🕶 🖬 🕶 🚳	@ \$₽000//!@%П•□•	
Meshing	General	1: Mesh •
Mesh Generation	Mesh	AND
Solution Setup	Scale Check Report Quality	
Models	Display	
Materials Phases	Solver	
Cell Zone Conditions	Type Velocity Formulation	
Mesh Interfaces	Pressure-Based Absolute Density-Based Relative	
Reference Values		
Solution	Steady Steady Planar	
Solution Methods Solution Controls	Transient Axisymmetric Axisymmetric Swirl	
Monitors Solution Initialization	Gravity Units	
Calculation Activities		
Results	Help	
Graphics and Animations		
Reports		ANSYS Fluent 14.5 (2d, dp, pbns, lam)
		writing default-interior (type interior) (mixture) Done.
		writing zones map name-id Done.
		Domain Extents: x-coordinate: min (m) = -5.0000000e+00, max (m) = 5.000000e+00 y-coordinate: min (m) = -5.000000e+00, max (m) = 5.000000e+00 Volume statistis: minimum volume (m): 1.2017At=-01 total volume (m): 1.2017At=-01 total volume (m): 1.2017At=-01 face area statistis: minimum face area (m2): 2.600604e-04 maximum face area (m2): 4.821094e-01 Checking mesh.

5.4. Solution Setup > Models > Viscous – Laminar > Edit... Choose the options below and click Ok.

B:Fluent Fluent [2d, dp, pbns, lam] [ANSYS Academic Teaching Advanced]					
File Mesh Define Sol	ve Adapt Surface Display Report Parallel Vi	ew Help			
: 📖 : 🎽 🕶 📓 🕇 🚳	@∥ऽ⊉Չ€↗!Չँ∏▾□▾				
File Mesh Define Sol Meshing Meshing Meshing Mesh Generation Solution Setup General Drame Define Cel Zone Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Solution Methods Solution Methods Solution Methods Solution Methods Solution Activities Run Calculation Results Graphics and Animations Plots Reports	Ver Adapt Surface Display Report Parallel Vis Weither Constraints of the second secon	Help Viscous Model Model Inviscid Laminar Spalart-Almaras (1 eqn) & kepsilon (2 eqn) Konega (2 eqn) Transition k44 onniga (3 eqn) Transition k44 onniga (3 eqn) Scale-Adaptive Simulation (SAS) Kepsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Frunctions Non-Equilibrium Wall Functions Non-Equilibrium Wall Functions Scaleabel Wall Frunctions Denhanced Wall Treatment User-Defined Wall Functions Enhanced Wall Treatment Options	Model Constants Cinu 0.09 C)-Epsion 1.44 C2-Epsion 1.92 TXE Prandt Number 1 User-Defined Functions Turbulent Viscosity none Prandt Numbers TXE Prandt Number		
		Options Curvature Correction OK	Cancel Hep		

5.5. Problem Setup > Materials > Fluid > air > Create/Edit. Change Density and Viscosity to experimental values and click Change/Create then click close.

Problem Setup	Materials	Create/Edit Materials		×
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh	Naterials Fruc Sold aluminum	Name air Chemical Formula	Material Type Fuld Fuld Materials FullENT Fluid Materials FullENT Fluid Materials Full Noture none	Order Materials by Name Chemical Formula FLUENT Database User-Defined Database
Retrence Values Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots		Properties Density (kg/m3) 1.2089 Viscosity (kg/m-4) 1.815e-05	• Edt	~
Reports	Create/Edt) Delete		Change/Croate Delete Close Help	

5.6. Solution Setup > Boundary Conditions > inlet > Edit. Change velocity to experimental condition and rest of the parameters to values shown below and click OK.

Problem Setup	Boundary Conditions	Velocity Inlet
General Models Materials	Zone airfoil default-interior	Zone Name inlet
Phases Cell Zone Conditions	inlet outlet	Momentum Thermal Radiation Species DPM Multiphase UDS
Mesh Interfaces		Velocity Specification Method Components
Reference Values		Reference Frame Absolute
Solution Solution Methods		Supersonic/Initial Gauge Pressure (pascal)
Solution Controls Monitors		X-Velocity (m/s) 7.04 constant
Solution Initialization Calculation Activities		Y-Velocity (m/s) 0 constant
Run Calculation		Turbulence
Results	Phase Turne ID	Specification Method K and Epsilon
Plots Reports	mixture velocity-inlet 5	Turbulent Kinetic Energy (m2/s2) 0.08 constant
	Edit Copy Profiles	Turbulent Dissipation Rate (m2/s3) 7,4 constant
	Parameters Operating Conditions	
	Periodic Conditions	OK Cancel Help

5.7. **Problem Setup** > **Boundary Conditions** > **outlet** > **Edit**. Change turbulence parameters to values shown below and click **OK**.

Problem Setup	Boundary Conditions	Pressure Outlet
Problem Setup General Models Materials Phases Cell Zane Conditions Extractast Central store Mesh Thiter faces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Controls Monitors Solution Activities	Zone arfal default-interior intet carlet	Zone Name Outlet Outlet Momentum Thermal Radiation Species DPM Multiphase UDS Gauge Pressure (pasced) 0 constant • Backflow Direction Specification Method Normal to Boundary • • Target Ness Fion Rate Turbulence Specification Method Internsity and Length Scale • Backflow Turbulent Intensity (Nb) 1,25 Specification • •
Results Graphics and Animations Plots Penorts	Phase Type ID mixture v pressure-outlet v 4	Backflow Turbulent Length Scale (m) 0.0035
	Edit Copy Profiles	OK Cancel Help
	Parameters Operating Conditions Display Mesh Periodic Conditions Help	

5.8. **Solution Setup** > **Reference Values**. Change reference values to the experimental values.

Problem Setup	Reference Values	
General Models	Compute from	•
Matenals Phases	Reference Values	
Cell Zone Conditions Boundary Conditions	Area (m2)	0.3048
Mesh Interfaces Dynamic Mesh	Density (kg/m3)	1.2089
Reference Values Solution	Depth (m)	1
Solution Methods Solution Controls	Enthalpy (j/kg)	0
Monitors Solution Initialization	Length (m)	1
Calculation Activities Run Calculation	Pressure (pascal)	0
Results Graphics and Animations	Temperature (k)	288.16
Plots Reports	Velocity (m/s)	7.04
	Viscosity (kg/m-s)	1.8152-05
	Ratio of Specific Heats	1.4
	Reference Zone	
		•
	Help	

5.9. Solution > Solution Methods. Change the option as per below.

Problem Setup	Solution Methods
General Models Materials Phases Cel Zone Conditons Boundary Conditons Meah Interfaces Dynamic Meah Reference Values Solution Solution Letherop Solution Centrols Monitors Solution Initialization Caldadato Activities	Pressure Helocity Coupling Scheme SpaRuE SpaRuE Cracetta Gradent Gradent Gradent Gradent Standard Versure Standard Vomentum Second Order Lowind Vomentum Second Order Lowind Vomentum Second Order Lowind Vomentum Voment
Run Calculation	Turbulent Dissipation Rate
Results	Second Order Upwind
Graphics and Animations Plots Reports	Transent Formulation Violation Viola

5.10. Solution > 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.

Problem Setup	Solution Controls	
General Models Materials Phases	Under-Relaxation Factors Body Forces 1	^
Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Momentum 0.5 Turbulent Kinetic Energy 0.5	ĺ
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Turbulent Dissipation Rate 0.5 Turbulent Viscosity 1	E
Results Graphics and Animations Plots Reports	Default Equations Limits Advanced	•

5.11. Solution > Solution Monitors > Residuals – Print, Plot > Edit... Change convergence criterions and click OK.

-					
Problem Setup	Monitors	Residual Monitors			— × —
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Residual's Statistic and Force Monitors Estatistic - Off J. Mol. Drag - Off Uff - Off Moment - Off Edit Surface Monitors	Options I on the Console Vindow I or Curves Axes Iterations to Flot 1000	Equations V-velocity k epsilon V V V V V V V V V V V V V	V V V	1e-5 1e-5 1e-5 1e-5
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots	Create) Edt (Delete) Volume Monitors	Iterations to Store	Residual Values Normalize Scale Compute Local Scale Renormalize	Iterations 5 v Cancel Hel	Convergence Criterion
Reports					

5.12. **Solution** > **Solution Initialization**. Change the velocity to experimental value and rest of the parameters as per below and click **Initialize**.

Problem Setup	Solution Initialization
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh	Intialization Methods Utilialization Standard Intialization Compute from Reference Frame
Reference Values Solution Solution Methods Solution Controls	Relative to Cell Zone Absolute Initial Values
Monitors Solution antivities Calculation Activities Run Calculation Results Graphics and Animations Pilots Reports	Gauge Pressure (pascal) * 0 X velocity (m/s) 7.04 * 7.04 * 0 Turbulent Kinetic Energy (m2/s2) 0.08 * Turbulent Kinetic Energy (m2/s2) * 7.4 *
	Initiaize Reset Patch Reset DPM Sources Reset Statistics

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



5.14. File> Save Project File> Name "Fluids CFD Lab 2"> Select zip file on H: Drive.

🖸 B	Fluent F	luent [20	d, dp, pb	ns, ske
File	Mesh	Define	Solve	Adapt
	Refresh	Input Da	ta	ĺ
	Save Pro	oject		
	Read			•
	Write			- 1
	Import			•
	Export			+
	Solution	n Files		
	Interpol	ate		
	EM Map	oping		- +
	FSI Map	ping		- +
	Save Pic	ture		
	Data Fil	e Quantit	ies	
	Close Fl	uent		

5.15. Close Fluent.

6. Continue Layout

6.1. In the workbench home screen, right click the **Fluent** bar on the **Fluent** component and select **Duplicate**.

A Fluids CFD Lab 2 - Workbench File View Tools Units Extensions H	telp	
New 📂 Open 🛃 Save 🔣 Save As	👔 Import 🖏 Reconnect 🖉 Refresh Project 🦩 Update Project 🔄 Project 🚯 Compact M	ode
Toolbox 🔻 🕂 🗙	Project Schematic	▼ ₽ X
Analysis Systems		
O Design Assessment ID Exctrc Deplict Dynamics Explict Dynamics Idiud Flow - Blow Molifon (Polyflow) E Fluid Flow - Extrusion (Polyflow) Fluid Flow (CFX) Idiud Flow (CFX) E Fluid Flow (Polyflow) E Idiud Flow (Polyflow) E Fluid Flow (Flown) E Idiud Flow (Flown) E Idiud Flow Flow Flow E Idiud Flow Flow Flow Flow Flow E Idiud Flow Flow Flow Flow Flow Flow Flow Flow	A 1 Wesh 2 Wesh 2 Winported Mesh ✓ 0-course:R=5-aoa=0 File Clear Generated Data Clear Generated Data Clear Generated Data Clear Generated Data Clear Generated Data	Ī
100 Magnetostatic 110 Modal	Properties	-
Random Vibration	Add Note	
📶 Response Spectrum		
Rigid Dynamics		
Static Structural		
T Steady-State Inermal		
View All / Customize		
Orag a Toolboxitem on top of a system to reus	se components and exchange data. 🗰 Show Progress 💭 Show 5 M	1essages

6.2. Drag and drop another **Mesh** component into the Workbench **Project Schematic**.

🔥 Fluids CFD Lab 2 - Workbench	
File View Tools Units Extensions I	telp
🎦 New 对 Open 🛃 Save 🔣 Save As	👔 Import 🛛 🖗 Reconnect 刘 Refresh Project 🛛 🗡 Update Project 💮 Project 🕜 Compact Mode
Toolbox 🔻 👎 🗙	Project Schematic 👻 🕂 🗙
Steady-State Thermal	A
Thermal-Electric	
Transient Structural	▼ A ▼ B ▼ C
🔃 Transient Thermal	1 🥮 Mesh 1 🖸 Fluent 1 💶 Fluent
Component Systems	2 🥥 Imported Mesh 🗸 🛶 🛛 2 🎇 Setup 🗸 📌 2 鯼 Setup 🗸
🐽 Autodyn	O-course-R=5-apa=0 3 Solution ✓
CFX	Thurst Committee
🥏 Engineering Data	Fluent Copy of Fluent
🔅 External Connection	
External Data	
Finite Element Modeler	
Fluent	▼ D
Fluent (with TGrid meshing)	1 💓 Mesh
🥪 Geometry	2 🥪 Geometry 💡 🖌
🚸 ICEM CFD	3 🍘 Mesh 🙄 🗸
Mechanical APDL	
Mechanical Model	O-course-R=4-aoa=0
🥔 Mesh	
Microsoft Office Excel	*
View All / Customize	•
🔋 Ready	🚥 Show Progress 💭 Show 5 Messages 🛒

6.3. Delete the connection between the original mesh component and the duplicated Fluent component by right clicking the line, selecting **Delete**, and clicking **OK**. Then reconnect the new mesh to the duplicated Fluent component. It should look similar to the layout below.



- 6.4. The new mesh can then be imported and renamed as per section 4. (Note: This is an efficient way to copy the Fluent setup. This can be done for all the remaining simulations that need to be run. This saves time from having to repeat the setup process for every simulation.)
- 6.5. The final layout should look similar to the layout below. (You should take a screen shot of your layout and add to the final report.)



7. Continuing Setup

7.1. Right click the duplicated Fluent **Setup** and click **Edit...**

🔥 Fluids CFD Lab 2 - Workbench		×
File View Tools Units Extensions H	telp	
🎦 New 对 Open 🛃 Save 🔣 Save As	👔 Import 🖗 Reconnect 🛿 Refresh Project 🦩 Update Project 🌀 Project 🕜 Compact Mode -	
Toolbox 🔹 🕂 🗙	Project Schematic	φ x
A Linguite A Linguite Linear Buckling Modal Random Vibration Response Spectrum Rigid Dynamis Static Structural Static Structural Transient Thermal Component Systems Audoyn CrX Eigineering Data External Connection External Data Finite Element Modeler	A 1 Witch 2 Proted Mitch 0 rautomatic course R=5-aa=0 1 Ruent 1 Ruent 2 Ruent 1 Ruent 2 Ruent 1 Ruent 2 Ruent 1 Ruent 2 Ruent 1 Ruent 2 Ruent	
Yiew All / Customize	Transfer Data To New	
Right-click to update component.	🧭 Update	5 . ::

7.2. Select Yes.



7.3. Select OK.



7.4. Solution > Solution Initialization > Click Initialize.

D:Copy of Fluent Fluen File Mesh Define So	t [2d, dp, pbns, ske] [ANSYS Academic Teachin Ive Adapt Surface Display Report Para	g Advar allel Vi	eed]
i 📖 i 📸 🕶 🛃 🕶 🔟	❷ 🕄 🔂 @ 🗶 🖉 🔍 🕄 📲 - 1	•	
Meshing	Solution Initialization		1: Mesh
Mesh Generation Solution Setup General Models Materials Discos	Initialization Methods Hybrid Initialization Standard Initialization Compute from		
Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Reference Frame Relative to Cell Zone Absolute	•	
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Initial volues Gauge Pressure (pascal) 0 x Velocity (m/s) 7.04	^	
Results Graphics and Animations Plots	Y Velocity (m/s) 0 Turbulent Kinetic Energy (m2/s2)]	Mesh Jul 26, 2013 ANSYS Fluent 14.5 (2d, dp, pbns, ske)
Reports	0.08 Turbulent Dissipation Rate (m2/s3) 7.4]	Setting zone id of default-interior to 7. Done. Setting fluid (nixture) Done. Setting infoil (nixture) Done. Setting outlet (nixture) Done. Setting default-interior (mixture) Done.
	Initialize Reset Patch Reset DPM Sources Reset Statistics	Ŧ	Done. Preparing mesh for display Done. Satting Post Processing and Surfaces information Done
	Help		v m v

- 7.5. Solution > Run Calculation > Click. (This method can be used for running the remaining simulations.)
- 7.6. File > Save Project.

D:Copy of Fluent Fluent [2d, dp, pbns, ske] [ANSYS Academic Teaching Advanced]						
	◎ ⑤ ⑦ ④ ④ / ◎ ◎ 八 開 - □ -	eow fielp				
Meshing Mesh Generation	Run Calculation Check Case Preview Mesh Motion	I: Mesh				
General Moterials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods	Number of Iterations Reporting Interval 10000					
Solution Controls Monitors Solution Initialization Calculation Activities RuniCalculation Results Graphics and Animations Plots Reports	Hep	Mesh Jul 26, 2013 ANSYS Fluent 14.5 (2d, dp, pbns, ske) Setting zone id of default-interior Done. Setting fluid (mixture) Done. Setting airfoil (mixture) Done. Setting inlet (mixture) Done. Setting default-interior (mixture) Done. Preparing mesh for display Done.				

8. Post Processing

Displaying Residuals and Mesh

Solution > Solution Monitors > Residuals - Print, Plot > Edit > Plot.

Problem Setup	Monitors	Residual Monitors				×
General Models Materials Phases Cell Zone Conditions Baundary Conditions	Residuals, Statistic and Force Monitors Residuals - Print, Plot Statistic - Off Urit - Off Moment - Off	Options Print to Console Plot Window	Equations Residual Continuity x-velocity	Monitor Check Convergence	e Absolute Criteria	* E
Mesh Interfaces Dynamic Mesh Reference Values Solution	Edit Surface Monitors	Iterations to Plot	v-velocity	V V V	1e-05	
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Create Edt Delete	Iterations to Store	Residual Values	Rerations	Convergence Crite absolute	erion v
Results Graphics and Animations Plots Reports	Volume Monitors	OK PI	Renormalize	Cancel He	-tp	

 $\label{eq:File} File > Save Picture. \ Using option \ as \ per \ below \ save \ figure.$

2 A	Fluid Flow (FLUENT) FLUEN	[2d, dp	o, pbns, ske] [AN	SYS Academic Teac	hing Introduct	ory]	
rile	Refresh Input Data	apt S	Save Picture	Report Parallel	view Help		8
	Save Project	Fo	rmat	Coloring	File Type	Resolution	
	Read Write		D EPS D JPEG D PPM	Color Gray Scale	Raster Vector	Width 960	
	Import Export		PostScript TIFF PNG	Options	Win	dow Dump Command	ē
	Solution Files Interpolate		Window Dump	V Landscape On	und im	port -window %w	
	EM Mapping		Save	Apply P	Clo	se Help	
	FSI Mapping						
	Save Picture						
	Data File Quantities						
	Close FLUENT		dit Delete				

Display > **Mesh** > **Display**.



Printing Forces

Results > **Forces** > **Setup** > **Print**. This will print the drag coefficient as per below.

Problem Setup	Reports			Force Reports		
General Models Materials Phases Cell Zone Conditions Beach Interfaces Dynamic Meth Solution Solution Methods Solution Institution Controls Solution Institution Controls Solution Institution Controls Run Calculation Run Calculation Results	Reports Floxes Projected Area Surface Integr Usare Integr Sampion - Usare Prose Sampion - Usare Pro- Sampion - Usare Pro- Sampion - Usarea Hatographic - Usa	Repots Forces Forces Projected Areas - Unavailable Sortines Integrate Districts Plates: Sample Hetotyom - Unavailable Hete Exchanger - Unavailable		Options Direction Vector Wall Zones IB Image: Second Seco		
Plots Reports	Help	arameters				1e-05 1e-06
Forces - Direction Vector Zone alrfoil Net	(1 0 0) Forces (n) Pressure 0.056012188 0.056012188	Uiscous 0.13746585 0.13746585	Total 0.19347804 0.19347804	Coefficients Pressure 0.0061342558 0.0061342558	Viscous 0.015054772 0.015054772	Total 8.021189028 8.021189028

Results > **Forces** > **Set Up...** Change the direction vector as per below and click **Print**. This will print the lift coefficient as per below.

Net	3.	0248787	0.0044405828	3.0293193	0.33127396	0.00048631685	0.3317602
Forces - Dir Zone airfoil	ection Vector (0 Fo Pr 3.	1 0) hrces (n) essure 0248787	Viscous 0.0044405828	Total 3.0293193	Coefficients Pressure 0.33127396	Viscous 0.00048631685	Total 0.3317602
	Calculation Activities Run Calculation Results Graphics and Animation Plots Reports	is Set Up Pi Help	arameters		Print Write	Close Help	1e-05
	Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization	Projected Area Surface Integra Volume Integra Discrete Phase Sample Histogram Summary - Ur Heat Exchange	is - Unavailable als :: :available ir - Unavailable		Wall Name Pattern Wall Name Pattern Save Output Parameter		
	General Models Materials	Reports Fluxes			Options Dire Forces X	o Wall Zor	xes 🗏 🚍
		Peporto		(Force Reports		— ×

Plotting Results

Results > Plots > XY Plot > Set Up... Select parameter as per below and click Plot.

Problem Setup	Plots	Solution XY Plot		×
General Models Materials Phases Cell Zone Conditions Boundary Conditions Meth Interfaces Dynamic Metho Rafference Values Solution Solution Methods Solution Controls Monitors Solution Controls Monitors Raff Calculation Raff Calculation Raff Calculation	Piota National File Profile Data - Unavailable Interpolated Data PT	Options Vestion on X Axis Position on Y Axis Position on Y Axis Write to File Order Points File Data	Plot Direction X 1 Y 0 Z 0 Load File, Free Data	Y Anis function Pressure
Graphics and Animations	Set Up	Plot	Axes	Curves] Close Help

Click Load File... and load the experimental pressure coefficient then click Plot.



Plotting Contours and Vectors

Results > **Graphics and Animations** > **Graphics** > **Contours** > **Setup**. Checked **filled**, select **static pressure** and click **Display**.



Results > **Graphics** and **Animations** > **Graphics** > **Vectors** > **Set** Up. Click **Display**.

Problem Setup	Graphics and Animations	Vectors	
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Graphics Mesh Contours Vectors Pathlnes Pathles Pathles	Options Global Range Auto Range Clip to Range V Auto Scale Draw Mesh	Vectors of Velocity Color by Velocity Velocity Magnitude
Dynamic Mesh Reference Values Solution	Set Up	Style arrow	Min (m/s) Max (m/s) 0.01338798 8.992969
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results	Animations Exemptions Source Animation Solution Animation Playback	Scale Skip 1 0 • Vector Options Custom Vectors	Surfaces E = airfoil default-interior inlet outlet
Graphics and Animations Plots Reports	Set Up Optons Scene Wews Lights Colormap Annotate	Surface Name Pattern Match	New Surface Surface Types axis dip surf exhaust-fan fan v
	Help	Display	iompute Close Help



Results > **Graphics and Animations** > **Graphics** > **Contours** > **Set Up...** Select parameters as per below and click **Display**. You can modify min and maximum to get a better figure.





9. C-Domain

9.1. From the Workbench home screen, right click on the C-Domain Study module's **Geometry** and select **New Geometry**.



- 9.2. Select Meter and click OK.
- ANSYS Workbench
- 9.3. File > Import External Geometry File... Select intro-airfoil.igs and click Open. Click Generate.
- 9.4. Add a new plane by selecting the **New Plane** button. For the **Type** select **From Point and Normal**.

🗱 E: C-Domain Study - Desi	gnModeler		
File Create Concent To	ols View Help		
	muo (r Redo Selecti		
ા જ્યસ્થ્ય	Q Q 38 X 100 •	12	
•••• •• •	5 / · / · / F		
XYPlane 💌 🦾 🗈	None 🔻 💯] 🕏	Generate 🛯 🖤 Share Topology 🔣 Parameters	
Extrude Revolve	Sweep 🔥 Skin/Loft	Thin/Surface 💊 Blend 👻 💊 Chamfer 🛳 Slice 📗 🚸 Point 😩 C	Conversion
Tree Outline		Graphics	4
E: C-Domain Study	r		
XYPlane			ANSYS 🛛
			R14.5
			Academic
⊞ √ 1 Part, 1 Body			
Sketching Modeling			
Details View	¢.		
 Details of Plane5 			
Plane			
Туре	From Point and Normal		
Base Point	Not selected		
Normal Defined by	Not selected		
Transform 1 (RMB)	None		
Reverse Normal/Z-Axis?	No		Y
HIP XY-Axes?	NO		
Export Coordinate System?	NO		•
		0.0 <u>00 0.1</u> 00 (m)	📕 🕹 🕹 🕹
		0.050	
		Model View Print Preview	
🦻 Plane Creation Choo	ose property to edit or click (Senerate to create the plane No Selection	Meter 0 0

9.5. For the Base Point, zoom in and select the point at the trailing edge as seen below.



9.6. For the **Normal Defined By**, select the **XYPlane** on the **Tree Outline**. This creates a plane with the origin at the trailing edge point.

🗱 E: C-Domain Study - Des	gnModeler		Internation of Chicag Chica, Marcad Real	- 6 <mark>- × -</mark>
File Create Concept To	ools View Help			
ເຊັ້ນ 🛛 🖉 🖉	Jndo @Redo Select: * 🗽 🍡 🕅	MM	A ⊕ @ @ Q Q X	
- · · /· /· /	5- h- h- # #			
XVPlane • 35	None 🦷 🎽 Generate 📾	Share Topology R Parameters		
Estrude Bevolve	Suren A Skin/Loft DThin/Surfz	ce Blend - Chamfer Slice	Point E) Conversion	
Tree Outline		1	Graphier	
E: C-Domain Stud	1		origines	
				ANSYS
ZXPlane				R14.5
YZPlane				Academic
Planef				
F- 1 Part, 1 Body				
				<u>4</u>
Sketching Modeling				
Dataile View				
Details of Plane6		*		
Plane	Plane6			
Туре	From Point and Normal			
Base Point	Vertex	[]		
Transform 1 (RMB)	Apply	Cancel		v
Reverse Normal/Z-Axis?	No			
Flip XY-Axes?	No		🖊 🖬	T a l
Export Coordinate System	No			•
				↓ ×
			0 0.0002	0.0004 (m)
			0.0001 0.000	3
			0.001 0.001	
			Model View Print Preview	· · · · · · · · · · · · · · · · · · ·
Plane Creation - Defi	the normal or click Generate to complete	the plane	1 Diane	Meter 0 0
- Plane Creation Den	The the normal of click deherate to complete		1 Plane	Meter 0 0
🥶 📄 🄇			the second se	▲ III III III III III III III III III I

- 9.7. Click Generate.
- 9.8. Make sure the plane you just created is selected, then click the New Sketch button.
- 9.9. **Sketching > 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.



9.10. Sketching > Rectangle by 3 Points. Draw a rectangle as per below.



9.11. **Dimensions** > **General**. Size the arc and rectangle with a radius of 5m and a width of 5m respectively as seen below.



- 9.12. Delete the line that makes the left side of the rectangle by selecting it and pressing **Delete** on the keyboard.
- 9.13. **Concept** > **Surface from Sketches**. Select the sketch you just made click **Apply**, the click **Generate**.
- 9.14. Create > Boolean. Make sure the Operation is set to Subract, for the Target Body select the domain and for the Tool Bodies select the airfoil by selecting the Surface Body under the Tree Outline which corresponds to the airfoil. Click Apply when selecting both bodies and then click Generate.
- 9.15. **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.**

- 9.16. 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.
- 9.17. Repeat step 9.16 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.
- 9.18. Split the vertical line from the rectangle in half as well.
- 9.19. Use the **Line From Points** in the **Concept** drop down menu to draw lines from the domain perimeter to the perimeter of the airfoil always starting from the domain and ending at the airfoil. 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.



9.20. Tools > Projection. 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.



- 9.21. File > Save Project. Close window..
- 9.22. From the Workbench home screen right click on **Mesh** and select **Edit...**
- 9.23. Right click on mesh > Insert > Mapped Face Meshing. Select all six surfaces and click **Apply.**
- 9.24. Right click on **Mesh > Insert > Sizing.** Selct lines below and click **Apply.** Change parameters as per below.



9.25. Right click on **Mesh > Insert > Sizing.** Selct the line below and click **Apply.** Change parameters as per below.

😭 E : C-Domain Study - I	feshing [ANSYS Academic Teaching Advanced]		1 4 7 9 8 8	Internation of C	Characterization in some the	- 6 - X
File Edit View Units	Tools Help 🛛 🕶 ಶ Generate Mesh 🏥 👪 🗚					
🗣 🦞 🖬 • 🖒 •	R R R 8- 5 + Q Q Q Q	Q Q 22 /2 @ B % T+				
E Show Vertices		A . A . Thicken Annotations	👍 📕 Random Colors 🐼 Annotation Preferen	sces		
Mesh 😴 Update 👘	Mesh - R Mesh Control - Ju Metric Graph					
Outline						
Filter Name -	R all	Edge Sizing 2				ANSYS
Project		8/5/2013 4:42 PM				R14.5
B = ∰ Hodel (3)	n Budy de de de de gene de Nordeng de Nordeng de Nordeng ang 2	Edge Samg 2			•	
D - 1 - 1 - 1 - 1 - 21	#1.1					
Scope	sizing	*				
Scoping Method	Geometry Selection					
Geometry	1 Edge					
Definition						
Suppressed	No					Y
Type	Number of Divisions					
Number of Divisions	100					T -
Rias Type	manu					•
Bias Option	Bias Factor	-				· · · · · · · · · · · · · · · · · · ·
Bias Factor	1000.		0.000	3.000	6.000 (m)	
			1.500	4.5	00	
		Country (Distance) in the				
		Geometry / Print Preview A Report Pr	eview/			
		Messages				÷
		Text		Association		Timestamp
Press F1 for Help		0 No Messages	No Selection		Metric (m kn N s V A)	Degrees rad/s Celsius
		Control messages			,	442.014
🥶 🔚 (C 🔍 🗖 🚺 🔛 🕻		and the second sec			▲ P 🗊 🕩 492.00M 8/5/2013

9.26. Right click on **Mesh > Insert > Sizing.** Selct the line below and click **Apply.** Change parameters as per below.



9.27. Right click on **Mesh > Insert > Sizing.** Selct the lines below and click **Apply.** Change parameters as per below.



9.28. Right click on **Mesh > Insert > Sizing.** Selct the line below and click **Apply.** Change parameters as per below.



9.29. Right click on **Mesh > Insert > Sizing.** Selct the lines below and click **Apply.** Change parameters as per below.



9.30. Right click on **Mesh > Insert > Sizing.** Selct the lines below and click **Apply.** Change parameters as per below.



9.31. Right click on **Mesh > Insert > Sizing.** Selct the lines below and click **Apply.** Change parameters as per below.



9.32. Right click on **Mesh > Insert > Sizing.** Selct the line below and click **Apply.** Change parameters as per below.



9.33. Right click on **Mesh** > **Insert** > **Sizing.** Selct the line below and click **Apply.** Change parameters as per below.

E : C-Domain Study - Meshing [ANSYS Academic Teaching Advanced]			Instantia di Citta	of Line Manual And	- 0 - X
File Edit View Units Tools Help 🕂 🥵 Generate Mesh 🏄 🐼 🗛 👰 🔻	Worksheet is				
		attack at Mandam Colors (2) America	D(
D show vertices @ wiretrame in cage Coloring • A • A • A • A • A	Thicken Annotations	riviesn 🐥 💼 Kandom Colors 🥑 Annotati	on Preferences		
Mesh 🗦 Update 🍘 Mesh 🔻 🍕 Mesh Control 🔻 🔐 Metric Graph					
Outline	4				ANCVC
Filter: Name 💌 🔮 🕢 🕀	8/5/2013 5:06 PM				ANJIJ
iii) myset → iii (Hed (U)) → iii (Hed	Edge Sting L0				Academic
Details of "Edge Sidna 10" - Sidna					
Cone					
Scoping Method Geometry Selection					
Geometry 1 Edge					
E Definition					
Suppressed No					
Type Number of Divisions					i i i i i i i i i i i i i i i i i i i
Number of Divisions 30					Ť
Behavior Hard					
Dias Type	-				×
Bias Factor 15.		0.000	0.050	0.100 (m)	• -
	\Geometry ⟨Print Preview⟩ Re	port Preview	0.025 0.075		
	Messages				å x
	Text		Association	Timestamp	
Deers Et for Help	O No Marra	ner No Selection		Metric (m kn N c V A) Degrees rad/	Calcius
	Ve No Mesa	yes pro selection		metric (m, kg, m, v, V, A) Degrees rad/s	LOCOM
🚳 📄 😂 🕙 🔼 🛝 😬 💟					▲ P* 12 () 500 PM 8/5/2013

- 9.34. Click on Mesh and under the Details of "Mesh" change the Physics Preference from Mechanical to CFD. Click **Generate**.
- 9.35. Select all the parts that make up the arc by holding down Ctrl and selecting them individually. Right click the selection and select Create Named Selection. Name this inlet.
- 9.36. Select the vertical line on the right side of the domain, right click it and Create Named Selection. Name this outlet.
- 9.37. Select the six faces and right click them, select Create Named Selections. Name them fluid.
- 9.38. Select the four curve that make the airfoil, right click and create named selections. Name them airfoil.
- 9.39. File > Save Project. Close Meshing window.
- 9.40. Repeat the steps used in other simulations for all remaining steps of CFD Process.

10. Verification and Validation (V&V Simulations Only)

- 10.1. From the workbench home screen, right click on the Fluent Solution and from the dropdown menu select edit...
- 10.2. Select File > Read > Journal...



10.3. Browse to the zip folder for lab 2, change Files of Type to All Files and select Final Lab 2 29 pt Journal and click ok.

Look in:	🔒 CFD Lab2	Final	•	3 🤌 📂 🛄 🗸	
æ	Name	*		Date modified	Туре
Recent Places	🔥 CFD Lab 2	! Template.wbpz 2 29pt Journal		7/26/2013 2:48 PM 7/26/2013 2:48 PM	ANSYS v File
Desktop					
Libraries					
Computer					
(100			
Network	•				
	Journal File	Final Lab 2 29pt Journal		L	ОК
	Files of hone:	All Film			Canaal

10.4. Click No. (To make sure the points were implemented properly, display the mesh and zoom in to count and verify there are 29 points on the airfoil surface as per below.)

Question	x
?	Does this journal file contain commands that read or write files after the calculation begins?
	Yes



10.5. 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 grids, which are used for V&V calculations. It is not needed for automatic grids.)

Solution XY Plot			×
Options Vode Values Position on X Axis Position on Y Axis Write to File Order Points	Plot Direction X 1 Y 0 Z 0	Y Axis Function Pressure Pressure Coefficient X Axis Function Direction Vector	•
File Data 🔳 🗏	Load File	Surfaces airfoil default-interior inlet point-1 point-10 point-11 point-12	
Write	Free Data	New Surface Curves Close Help	

- 10.6. Open the V&V Excel template from the zip file.
- 10.7. Copy and paste the pressure coefficients into the proper sheet corresponding to the grid size. To do this open the saved coefficient data in TextPad, use the "Ctrl + a" function to select all, then right click and select copy.

TextPad - C\Users\mconger\De	skton\Medium Grid Press Coeff (29 nts)			
Eile Edit Search View Te	aals Master Configure Window Help			
	los macros configure window nep	Contraction of Final Street	cromostallu 🛛 û 🗖 Matela e	240
	Madium Grid Deers Coeff (20 ats)		icrementary & [] [] match c	- X
Medium Grid Press Coeff (29 nts)				
	(labels "Position" "Pressure Coeffic	cient")		
	((xy/key/label "noint=1")			
	0 0.952393			E
		Properties		
	((xy/key/label "point-10")	Cut		
	0.12226 -0.562213	Сору		
		Paste		
	((xy/key/label "point-11")	Cut Other	F	
)	Copy Other	•	
		Insert	•	
	((XY/Rey/label "point-12") 0.18338 -0.405737	Delete	•	
		Change Case	•	
	((xy/key/label "point-13")	Transpose	>	
	0.21399 -0.300439	Align	+	
		Reformat		
	((xy/key/label "point-14") 0.24451 -0.165764	Block Select Mode		
		Fill Block		
	((xy/key/label "point-15")	Spelling		
Explored The Document of Clip L		Toggle Bookmark		
Courte Decelle		roggie debininini		
Search Results				¥ ^
St Search Perults 🕅 Teel Output				
Constitute and atting to the Clinks				
Copy the selection to the Clipboard				119 1 Reau Ovi Diock Sync Rec Caps

10.8. 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.

X	🔜 🤊 - (° -	Ŧ	_		Template	V&V for a	iirfoil with	data extraction.	dsx - Microsoft	Excel		_		_		- • ×
F	File Home Insert Page Layout Formulas Data Review View Custom Commands 🛆 🕜 🗆 🔂															
ľ	Calib	ri * 11 * A* A*	= = =	≫,-	📑 Wrap Te	t	General	÷	≤₹	🕎 🚽	+	*		Σ AutoSum	Ż	A
Pas	te 🚽 B	IU- 🖽 - 🔕 - 🗛 -	≡ ≡ ≡	ŧ e	Merge &	Center +	\$ - 9	/s • • • • • • • • • • • • • • • • • • •	Conditional	Format Cell	Insert	Delete F	Format	Clear y	Sort & F	ind &
Clip	board G	Font		Alianme	ent	5	N	umber 5	Formatting *	as Table * Styles *		Cells		Ec	filter * 5	elect *
_										~						
			D	0	D	-		C	1			V		M	N	
1	(title "Pressur	e Coefficient")	D	L.	U	E .	F	G		J		ĸ		- IVI	IN	-
2	(labels "Positi	on" "Pressure Coefficient")					V&V	Data (Copy and	d paste v valu	e into V&V tem	plate)					
3		,					Point	x coordinate	v coordinate	Pressure Coeffi	cient					
4	((xy/key/labe	Paste Options:					1	0	0.00000		0					
5		Ā	952393				2	0	0.00609		0					
6)						3	0	0.00939		0					
7		Keep Text Only (T)					4	0	0.01401		0					
8	((xy/key/labe	point-10")					5	0	0.01712		0					
9		0.12226	-0.562213				6	0	0.01969		0					
10)						7	0	0.02356		0					
11							8	0	0.02613		0					
12	((xy/key/labe	"point-11")					9	0	0.02818		0					
13		0.15282	-0.487262				10	0	0.02833		0					
14)						11	0	0.02674		0					
15	11						12	0	0.023/1		0					
10	((xy/key/labe	["point-12"]	0.405707				13	0	0.01932		0					
1/	1	0.18338	-0.405737				14	0	0.01371		0					
10	,	_					15	0	0.00753		0					
20	//w/key/labe	["noint-12")					17	0	-0.00122		0					
21	((A) AC I AC C	0.21399	-0.300439				18	0	-0.00315		0					
22)						19	0	-0.00421		0					
23	<u> </u>						20	0	-0.00545		0					
24	((xy/key/labe	l "point-14")					21	0	-0.00670		0					
25		0.24451	-0.165764				22	0	-0.00754		0					
26)						23	0	-0.00848		0					
27							24	0	-0.00871		0					
28	((xy/key/labe	l "point-15")					25	0	-0.00833		0					
29		0.27504	-0.029141				26	0	-0.00795		0					
30)						27	0	-0.00727		0					
31							28	0	-0.00596		0					
32	((xy/key/labe	l "point-16")					29	0	-0.00467		0					
33		0.27504	0.0845124													
34)		aking Life Cook	/ Eine	Caid Danas Ca	-66	a dia ma	d Dunne Coneff	Canada Crid	Dense Casti 0						
Pas	why Cequa	CONS VOLV VEIDULY VEIDU	acon the coer	rine	ond Press Co		eurum Gr	iu Press COEIT	Coarse Gho	Freas Coeff	0.1					

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

11. Exercises

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

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

11.1. **Effect of domain size:** Run 5 simulations using five different domain sizes using grids O-course-R5, O-course-R4, O-course-R3, O-course-R2 and O-course-R1. Fill the table 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 to be saved: None.

• Data to be saved: the above table with values.

11.2. Effect of numerical scheme on Verification study for lift coefficient and validation of pressure coefficient:

Use "O" type geometry with 0 degree angle of attack. 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 2^{nd} 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 grid uncertainty? Discuss the validation figure. For which locations of 29 points the CFD simulation has been validated? For which locations the CFD simulation has not been validated? 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 to be saved (only for the numerical scheme you used, but you must also present the figures for the calculations from your partner): 1. The "O" mesh you imported. 2. "Mesh Convergence" panel and "Verification" panel for lift coefficient. 3. Validation

figures for pressure coefficient. 4. Iterative history for lift coefficients on fine mesh.

• Data to be saved: None.

- 11.3. **"C" mesh generation:** Use "C" type domain and zero degree angle of attack for geometry and use the following parameters for mesh generation. Other parameters are the same as the values in the instruction part.
- Figures to be saved: "C" mesh generated by yourself.
- Data to be saved: converged lift coefficient.

- 11.4. **Effect of angle of attack on airfoil flow:** Using "O", automatic "coarse" meshes, run two simulations using angle of attack 0 degrees and 6 degrees, respectively. Analyze the difference of flow fields. Which case has a higher lift coefficient, which has a higher drag coefficient?
- Figures to be saved (for both attack angles): 1. pressure contours, 2. comparisons with EFD on pressure coefficient distribution, 3. velocity vectors near airfoil surface, 4. streamlines near the airfoil surface.
- Data need to be saved (for both attack angles): lift and drag coefficients.

11.5. **Questions need to be answered when writing CFD report:**

- 11.5.1. Answer all the questions in exercises 1 to 4
- 11.5.2. Analyze the difference between CFD/EFD and possible error sources.