# Simulation of Turbulent Flow over the Ahmed Body

#### 58:160 Intermediate Mechanics of Fluids CFD LAB 4

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 4 is to simulate **unsteady turbulent** flows over the Ahmed body following the "CFD process" by an interactive step-by-step approach and conduct verifications using CFD Educational Interface (FlowLab 1.2). Students will have "hands-on" experiences using FlowLab to **predict drag coefficients and axial velocity for slant angle 25 degrees and compare them with EFD data.** Students will use post-processing tools (streamlines, velocity vectors, contours, animations) to **visualize the mean and instantaneous flow fields and compute the non-dimensional shedding frequency (Strouhal number)**. Students will analyze the differences between CFD and EFD and present results in a CFD Lab report.



Flow Chart for ANSYS

## 2. Simulation Design

The problem to be solved is unsteady turbulent flows over the Ahmed body (2D). Reynolds number is around 768,000 based on inlet velocity and vehicle height (h). The following figure shows the sketch window you will see in FlowLab with definitions for all geometry parameters. The origin of the simulation is located at the rear of the body.  $\theta$  is the slant angle. L is the length of the body and h is the height of the body. Uniform velocity specified at inlet and constant pressure specified at outlet. The top boundary of the simulation domain is regarded as "Symmetry" and there is a distance between the car body and road, GL.



In CFD Lab4, all EFD data for turbulent airfoil flow in this Lab will be provided by the TA and saved on the Fluids Lab computers.

# 3. Open ANSYS Workbench Template

- 3.1. Download CFD Lab 4 Template from class website.
- 3.2. Open Workbench Project Zip file simply by double clicking file. This file contains all the systems that must be solved for CFD Lab 4.



# 4. Setup

4.1. Right click Setup and select Edit



4.2. Select double precision and click Ok

FLUENT Launcher (Setting Edit Onl	y) 🗖 🗖 📈 🖉
<b>ANSYS</b>	FLUENT Launcher
Dimension 2D 3D	Options Double Precision Use Job Scheduler
Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme Do not show this panel again	Processing Options Serial Parallel
Show More Options	Cancel Help -

## 5. Problem Setup

5.1. Problem Setup > General. Change solver to transient as per below.

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

5.2. Problem Setup > Models > Viscous > Edit. Change the turbulent model and near-wall treatment as per below.

Problem Setup	Models	Viscous Model	
General Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Models Multiphase -Off Precy - Off Viscoles Lamase Radiation - Off Heat Exchanger - Off Species -Off Discrete Phase - Off Solidification & Melting - Off Acoustics - Off Edit Help	Model Dinviscid Laminar Spalart-Almaras (1 eqn) K-expsilon (2 eqn) Transition k-4 omega (3 eqn) Transition k-4 omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) K-epsilon Model Standard Realizable Near-Wall Treatment Standard Wall Functions Non-Equilibrium Wall Functions Diser-Defined Wall Functions	Model Constants Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 User-Defined Functions Turbulent Viscosity none Prandtl Number TRE Prandtl Number
		OK	Cancel Help

5.3. Problem > Materials > Fluid > air > Create/Edit. Change the air density and viscosity as per below and click Change/Edit then close the window.

Problem Setup	Materials	Create/Edit Mate	erials			<b></b>
General Models Materials	Materials Fluid	Name		Material Type	•	Order Materials by Name
Phases Cell Zone Conditions Boundary Conditions	Solid aluminum	Chemical Formula		FLUENT Fluid Materials	•	Chemical Formula
Mesh Interfaces Dynamic Mesh Reference Values		Dreportion		Mixture none	Ψ.	User-Defined Database
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Create/Edit Delete	Properties Density (kg/m3) Viscosity (kg/m-s)	constant 1.225 constant 1.787e-05 Change/Create	Edt     Edt     Delete     Clo	se Help	

5.4. Problem Setup > Boundary Conditions > inlet > Edit. Change the inlet boundary conditions as per below and click OK.

Problem Setup	Boundary Conditions	Velocity Inlet	×
General	Zone	Zone Name	
Materials	ahmed_bottom ahmed_top	inlet	
Phases Cell Zone Conditions	back default-interior	Momentum Thermal Radiation Species DPM Multiphase UDS	
Boundary Conditions Mesh Interfaces	inlet nose	Velocity Specification Method Components	
Dynamic Mesh Reference Values	road	Reference Frame Absolute	<b>-</b>
Solution	upper-wall	Supersonic/Initial Gauge Pressure (pascal)	-
Solution Methods Solution Controls		X-Velocity (m/s) 40 constant	<b>-</b>
Monitors Solution Initialization		Y-Velocity (m/s) 0 constant	<b>_</b>
Calculation Activities Run Calculation		Turbulence	
Results		Specification Method K and Epsilon	-
Graphics and Animations Plots	Phase Type ID mixture v velocity-inlet v 11	Turbulent Kinetic Energy (m2/s2) 2.93047 constant	-
Reports	Edit Copy Profiles	Turbulent Dissipation Rate (m2/s3) 10 constant	-
	Parameters Operating Conditions		
	Display Mesh Periodic Conditions	OK Cancel Help	

5.5. Problem Setup > Boundary Conditions > Zone > outlet > Edit. Change the outlet boundary condition as per below and click OK.

Problem Setup	Boundary Conditions	Pressure Outlet
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Zone       ahmed_bottom       ahmed_top       back       default-interior       inlet       nose       outlet       road	Zone Name           Outlet           Momentum         Thermal         Radiation         Species         DPM         Multiphase         UDS         Gauge Pressure (pascal)         Constant <ul> <li>Backflow Direction Specification Method</li> <li>Momentum</li> <li>Registration</li> <li>Registration</li> <li>Registration</li> </ul> Figure 1     Registration     Image: Registration
Solution Solution Methods Solution Controls	slope upper-wall	Average Pressure Specification Target Mass Flow Rate Track Langer
Monitors Solution Initialization Calculation Activities Run Calculation		Specification Method Intensity and Length Scale
Results		Backflow Turbulent Length Scale (m)
Graphics and Animations Plots Reports	Phase Type ID mixture v pressure-outlet v 10	
	Edit Copy Profiles	OK Cancel Help

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

Problem Setup	Reference Values	
General Models	Compute from	•
Materials Phases	Reference Values	
Cell Zone Conditions Boundary Conditions	Area (m2)	0.288
Mesh Interfaces Dynamic Mesh	Density (kg/m3)	1.225
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	Temperature (k)	288.16
Plots Reports	Velocity (m/s)	40
	Viscosity (kg/m-s)	1.787e-05
	Ratio of Specific Heats	1.4
	Reference Zone	
		•
	Help	

#### 6. Solution

6.1. Solution > Solution Methods. Change solutions methods as per below.



6.2. Solution > Solution Controls. Change under-relax factors for pressure, momentum, turbulent kinetic energy, and turbulent dissipation rate to 0.7.

Problem Setup	Solution Controls
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Under-Relaxation Factors       0.7       Density       1       Body Forces       1
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Peoults	Momentum 0.7 Turbulent Kinetic Energy 0.7 Turbulent Dissipation Rate 0.7
Graphics and Animations Plots Reports	Default Equations Limits Advanced

6.3. Solution > Monitors > Edit. Change the convergence criterion and click OK.

Problem Setup	Monitors	Residual Monitors			
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Residuals, Statistic and Force Monitors Residuals - Print, Plot Statistic - Off Drag - Off Lift - Off Moment - Off Edit Surface Monitors	Options V Print to Console V Plot Window 1  Curves Axes Iterations to Plot 1000 V	Equations X-velocity V-velocity K epsilon V	v V V V	0.001 0.001 0.001 0.001
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Create Edit Delete	Iterations to Store	Residual Values Normalize Scale Compute Local Scale Renormalize	Iterations 5 (*) (*) (*) (*) (*) (*) (*) (*)	Convergence Criterion absolute

6.4. Solutions > Solution Initialization. Change x-velocity and turbulent parameters as per below.

Problem Setup	Solution Initialization
General Models Materials Phases Cell Zone Conditions Boundary Conditions	Initialization Methods           O         Hybrid Initialization           Image: Standard Initialization         Standard Initialization
Mesh Interfaces Dynamic Mesh Reference Values	Reference Frame  Relative to Cell Zone  Academic Science Scien
Solution Methods Solution Methods Solution Controls Monitors Solution Tritlatization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Initial Values         Gauge Pressure (pascal)         0         X Velocity (m/s)         40         Y Velocity (m/s)         0         Turbulent Kinetic Energy (m2/s2)         2.061034         Turbulent Dissipation Rate (m2/s3)         2620.743
	Initialize Reset Patch

6.5. Solution > Run Calculation. Change parameters as per below and click Calculate.

Problem Setup	Run Calculation
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh	Check Case       Preview Mesh Motion         Time Stepping Method       Time Step Size (s)         Fixed       0.0001         Settings       Number of Time Steps         1000       V
Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities <u>Run Calculation</u> Results Graphics and Animations Plots Reports	Options  Extrapolate Variables  Sampling Interval  Max Iterations/Time Step  Profile Update Interval  Data File Quantities  Calculate



#### 7. Results

7.1. Surface > Line/Rake. Create 10 lines at the locations given at the table below.

	Surface Display Report
	Zone
	Partition
	Point
	Line/Rake
	Plane
	Quadric
	Iso-Surface
	Iso-Clip
	Transform
	Manage
Line/Ra	ke Surface
Options	Type Number of Points
📃 Line To	
Reset	
End Points	
x0 (m) -(	0.26208 ×1 (m) -0.26208
y0 (m) 0	y1 (m) 2.95
z0 (m) 0	z1 (m) 0
	Select Points with Mouse
New Surfac	e Name
posidonei	
Creat	e Manage Close Help

Surface Name	x0	y0	x1	y1
Position-1	-0.26208	0.00	-0.26208	2.95
Position-2	-0.11200	0.00	-0.11200	2.95
Position-3	-0.06192	0.00	-0.06192	2.95
Position-4	-0.01209	0.00	-0.01209	2.95
Position-5	0.03801	-0.05	0.03801	2.95
Position-6	0.08812	-0.05	0.08812	2.95
Position-7	0.18806	-0.05	0.18806	2.95
Position-8	0.28800	-0.05	0.28800	2.95
Position-9	0.43800	-0.05	0.43800	2.95
Position-10	0.63790	-0.05	0.63790	2.95

7.2. Define > Custom Field Functions. Create custom field functions and click Define. You will need to create two custom field functions shown in the table below.

Defir	ne Solve	Adapt	Surface	Display
	General			
	Models			
	Materials			
	Phases			
	Cell Zone (	Conditio	ns	
	Boundary (	Conditio	ns	
	Operating	Conditio	ns	
	Mesh Inter	faces		
	Dynamic M	/lesh		
	Mesh Mor	pher/Opt	timizer	
	Mixing Pla	nes		
	Turbo Top	ology		
	Injections			
	DTRM Rays	s		
	Shell Cond	uction V	/alls	
	Custom Fie	eld Funct	tions	
	Parameters	5		
	Profiles			
	Units			

Custom Field Function Calculate	or 📃			
Definition y / 0.288 + - X / y^ INV sin cos tan lin 0 1 2 3 4 5 6 7 8 9 ( ) PI e . New Function Name y-by-h	x     ABS     Select Operand Field Functions from       h     log 10     Field Functions       K     K     V       SQRT     V     V       CE/C     Select       DEL     Select			
Function Name Definition				
y-by-h	y/0.288			
Mod. U	(mean-x-velocity/120)+(x/0.288)			

7.3. Results > Plots > XY Plot > Set Up. Click load file and load the experimental data. Select the lines you created (position-1 through position-10) and experimental data then click Plot.

Problem Setup	Plots	Solution XY Plot	×
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Plots IVY Plot Histogram File Profile: Profile: Interpolated Data FFT	Options         Plot Direction         Y Axis Function           Position on X Axis         I         I         I           Position on X Axis         Y         I         V         X Axis Function           Order Points         Z         I         I         I         I           File Data         I         I         I         I         I         I         I           Modified Velocity (C1VI)         Surfaces         Surfaces         Surfaces         Surfaces         Surfaces	ns •
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results		Load File	E V
Graphics and Animations	Set Up	Free Data New Surface 🔻	
Reports	Нер	Plot Axes Curves Close	Help

4.00e+00		1.1	T Î T	1	P	1	T.	- Alexandra
3.50e+00								
3.00e+00								
2.50e+00				:	:			
2.00e+00 —				:	:	:		
1.50e+00				i		1	1	
1.00e+00				j	1	1	j	
5.00e-01		5			/	1		
0.00e+00 -		•	•	in l	)	/	Ø	
-5.00e-01	-0.5	n	0.5	1	15	2	2.5	3
	210	1	210	mod.u	1.0	-	2.0	

Note: You change the style and color of the data by clicking Curves button and changing the parameters below then clicking apply. Adjust Y axis maximum to 2.5 and minimum to -0.5.

Curves	- Solution XY Plot	×				
Curve #	Line Style Pattern Color black Weight 1	Marker Style Symbol Color red Size 0.3				
ADDV. Close Help						

7.4. Results > Graphics and Animations > Contours. Change parameters as per below and click display.

Problem Setup	Graphics and Animations	Contours		×
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Graphics Mesh Contours Vectors Pathlines Particle Tracks Set Up	Options Filled Okade Values Global Range Clip to Range Draw Profiles Draw Mesh	Contours of Pressure Static Pressure Min (pascal) Max (pascal) [4096.963 [1334.687] Surfaces Surfaces	
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Animations Sweep Surface Scene Animation Solution Animation Playback	Levels Setup 100 • 1 • Surface Name Pattern Match	ahmed_bottom back default-interior inlet nose	-
Results Graphics and Animations Plots Reports	Set Up		Surface Types axis dip-surf exhaust-fan fan	
	Lights Colormap Annotate	Display	Compute Close Help	



7.5. Results > Graphics and Animations > Vectors > Set Up. Change parameters as per below and click display.

Problem Setup	Graphics and Animations	Vectors	
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh	Graphics Mesh Contours Vectors Pathlnes Particle Tracks	Options Ø Global Range Ø Auto Range Ø Auto Scale Draw Mesh Style	Vectors of           Velocity           Color by           Velocity           Velocity Magnitude           Min (m/s)           Max (m/s)
Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Blote	Set Up Animations Sweep Surface Scene Animation Solution Animation Playback	Arrow	0.04127333     90.05577       Surfaces     Image: Constraint of the second sec
Reports	Options       Scene       Views         Lights       Colormap       Annotate         Help       Help	Display	Surface Types E Cose Help



7.6. Results > Graphics and Animations > Contours > Set Up. Change parameters as per below and click Display.

Problem Setup	Graphics and Animations	Contours	×
General Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Meshods Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results <del>Calculation Activities</del> Reports	Graphics Mesh Contours Vectors Pathlines Particle Tracks  Set Up Animations Scene Animation Solution Animation Playback  Set Up	Options I filled V Global Range V Global Range V Global Range Oraw Profiles Draw Mesh 100  1 Surface Name Pattern Match	Contours of Velocity   Stream Function  Min (kg/s)  Max (kg/s)  Surfaces  ahmed jop back default-interior inlet rose  New Surface Types  Surface Types  Surface Types  Attrian  At
	Options Scene Views Lights Colormap Annotate	Display	Compute Close Help

7.7. Results > Reports > Forces > Setup. Change parameters as per below and click print.



> Forces	Former (p)						
Zone	Pressure			Viscous			Total
ahmed_boccom	(0 724.71536	6) 6)		(4.2038587 0 0) (4.6776174 0 0)			(4.2038587 -7.1230193 0) (4.6776174 724.71536 0)
back	(27.573758 0	0)		(0 0.021909745 0	1)		(27.573758 0.021909745 0
nose	(-16.074916 3	63.49982 0)		(2.4533813 1.701	7059 0)		(-13.621535 365.20153 0)
slope	(74.575871 15	9.92847 0)		(0.11512858 -0.0	15368534 0)		(74.691 159.87479 0)
Net	(86.074712 12	41.02 0)		(11.509986 1.669	9303 0)		(97.584698 1242.69 0)
Forces - Direction Vector	r (1 0 0) Farrans (n)			Coofficients			
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total	
ahmed_bottom	0	4.2638587	4.2638587	0	0.01510721	0.01510721	
ahmed_top	0	4.6776174	4.6776174	8	0.016573192	0.016573192	
back	27.573758	0	27.573758	0.097696141	8	0.097696141	
nose	-16.074916	2.4533813	-13.621535	-0.056954779	0.008692536	-0.048262243	3
slope	74.575871	0.11512858	74.691	0.26422858	0.00040791025	0.26463649	
Net	86.074712	11.509986	97.584698	0.30496994	0.040780848	0.34575079	

### 8. Exercises

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

### **Simulation of Turbulent Flow over the Ahmed Body**

- You can save each case file for each exercise using "file"→ "save as"
- Otherwise stated, use the parameters shown in the instruction.
- 8.1. Simulation of turbulent flows over Ahmed body (slant angle=25 degree):

Use 25 degrees for slant angle to create the geometry, create "**Tri Coarse**" mesh, and run the simulation with time steps 1400. **NOTE: This simulation could take up to 3 hours.** 

**a.** Fill in the table for the four drag coefficients and compute the relative error between CFD and EFD (Ahmed data), EFD data for  $C_k$ ,  $C_B$ , and  $C_s$  can be found from the figure below. Where  $C_k = C_k^*$ ,  $C_B = C_B^*$ , and  $C_s = C_s^*$ . The definitions of the drag coefficients are:  $C_k$  is the forebody pressure drag coefficient,  $C_B$  is the vertical based pressure drag coefficient,  $C_s$  is the slant surface pressure drag coefficient, and  $C_w = C_D$  is the total drag coefficient. So,  $C_w = C_D = C_S + C_B + C_k + C_R$ 



	$C_k$	C <sub>B</sub>	Cs	CD
Ahmed (EFD)				0.289
k-e				
Error (%)				

#### **b.** Questions:

• Do you observe separations in the wake region (use streamlines)? If yes, where is the

location of separation point?

• What is the Strouhal number based on the shedding frequency (C<sub>D</sub> vs. time), the height

of the Ahmed body and the inlet velocity? Note: the shedding frequency f=1/T where T

is the typical period of the oscillation of  $C_D$  that can be evaluated using the peaks between 0.1<time<0.14.

• **Figures to be saved:** 1. XY plots for residual history, axial velocity vs. x/h (with EFD), TKE vs. x/h and time history of drag coefficient, 2. Contour of pressure, contour of axial velocity and velocity vectors, 3. 3 or 4 snapshots of animations for

turbulent-viscosity-ratio and streamlines (hints: you can use <<**Alt+print Screen**>> during the play of the animations).

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