Riccardo Zamolo, Enrico Nobile

DIA - Dipartimento di Ingegneria e Architettura Università degli Studi di Trieste

Esercitazioni di Termofluidodinamica Computazionale

External aerodynamics over Ahmed's body with ANSYS Fluent



April 2020

1 Introduction and problem definition

This tutorial document shows how to perform a full CFD analysis of a vehicle aerodynamics problem using ANSYS Workbench 2020 platform (hereafter "WB").

The geometry will be defined using ANSYS SpaceClaim 2020 (hereafter "SpaceClaim"), the mesh will be generated using ANSYS Meshing 2020 (hereafter "ANSYS Meshing") and the problem will be solved using ANSYS Fluent 2020 (hereafter "Fluent") with RANS (Reynolds-Averaged Navier-Stokes) formulation: a turbulence model will be employed to find a steady-state solution.

For the sake of simplicity, the specific vehicle for this study case is the Ahmed's body [1], whose geometry is reported in Figure 1, in the case of a rear slant angle of 25° .

Beside its basic geometry, the physics of the problem is not trivial:

"Most of the drag of the body is due to pressure drag, which is generated at the rear end. The structure of the wake is very complex, with a separation zone and counter-rotating vortices coming off the slant side edges, whose strength is mainly determined by the base slant angle." [3] (See Figure 2).

Once the characteristics of the problem have been defined as in Table 1, where air at $T = 20^{\circ}$ C is the working fluid, our main goal is to compare the computed drag coefficient C_D to experimental data:

$$C_D = \frac{2F_x}{\rho u_\infty^2 A_x} \tag{1}$$

where F_x is the computed x-component of the force acting on the body (drag force) and A_x is the projected body area along the x-axis.

Since we'll use a RANS (time-averaged) approach, we can take advantage of the symmetry of the problem around the x-y plane (see Figure 1) for both geometry and boundary conditions to simulate half the space around the Ahmed's body.



Figure 1: Ahmed's body with a rear vehicle slant angle of 25° (dimensions in mm).



Figure 2: Rear separation zone (from [1]).

Cha	aract	teristics of the problem.
L	=	1044 mm
W	=	389 mm
H	=	288 mm
A_x	=	0.057516 m^2
u_{∞}	=	40 m/s
T	=	20° C
μ	=	1.789×10^{-5} kg/(m·s)
ho	=	1.225 kg/m^3
Re	=	$\frac{\rho u_{\infty} L}{\mu} = 2.86 \times 10^6$

Table 1: Problem characteristics.

2 Workbench project

- Start WB and drag&drop a *Fluid Flow (Fluent)* component into the main window as in Figure 3; rename it *Ahmed_body* and save this new project: *File* \rightarrow *Save as...*



Figure 3: WB project (Fluid Flow with Fluent).

2.1 Geometry definition with SpaceClaim

- Right click on *Geometry* and click on *New SpaceClaim Geometry*... to define a new geometry with SpaceClaim (Figure 3).



Figure 4: SpaceClaim.

- Click on *Sketch mode* icon \mathbb{Z} and move the mouse in the main window to position the sketch on the x - y plane (or click on the z axis), then click on *Plan view* icon Plan View to have an orthogonal plan view (Figure 4).

- Using *Line*, define the simplified 2D profile of the Ahmed's body reported in Figure 5:

🔊 - 연 - 📲 💼 🛋 🕅				
File Design Insert Detail D)isplay Measure Repair Prepare She	et Metal KeyShot		
Paste Clipboard	C	Put Move Fill & Combine & Project Edit Intersect Create	gent ⊗PRgid 1 ∰ Gear nt ↓ Anchor kssembly	
Structure a b b curves b curves	 Drag to draw a straight line. Or click to create each point of a polyline, then double-click to end the line. 	t to select a tangent chain. Double-click again to select a closed loop. Drag to edit yo	ur sketch.	ANSYS
	Press F1 for more help, or F3 for a video.			R18.0
		◄ 843mm ─────	- -	
Structure Layers Selection Groups Views				
Options - Selection	9			
II Sketch	8 <u>^</u>			
Maintain sketch connectivity				
1 Sketch	8			
Snap to grid]—	
Snap to angle	288mm			
Create layout curves	20011111		1	94mm
Propercies				
	*		v +	
			50mm	- × A
		- 1044mm —	^ 	
	(Å)=x			
Properties Appearance	∧ Design 1* ×			4 Þ X
Click to select. Double-click to select a tangent of	hain. Double-click again to select a closed loop. Drag	to edit your sketch. x=-891,9454 y=	16,5336 📔 🔺 🕴 🌲 🛧	• 🗆 • 🖗 • 🕂 🖉 🖉

Figure 5: Ahmed's body starting profile.



- Click on *View* icon **()** and select the *Trimetric* view (Figure 6):

Figure 6: Trimetric view.

- Now we can extrude the profile using the *Pull* command: click on the profile and insert 194,5mm by keyboard to pull the profile along the +z direction (Figure 7):



Figure 7: Pull command to extrude the profile.

- Use middle mouse button to rotate the view, mouse wheel to zoom the view, Ctrl + middle mouse button to move the view.

- To create the double roundings on the vehicle front, select *Pull* command and click on one of the front edges, then click on *Round* icon (Figure 8):



Figure 8: Roundings.

- Specify 100mm by keyboard as rounding radius (Figure 9):

💐 늘 🖷 ち・ペ・ 🔹	A:Ahmed_body - Design1 - SpaceClaim	- & ×
File Design Insert Detail	Display Measure Repair Prepare Sheet Metal KeyShot	
Image: Second secon	N □ O N 2 N × D Ø A Ø A Ø B <th></th>	
Structure	Create round from 1 edge	
▲ ⊴ 😭 Design1*	Image: Note of the second	ANSYS R18.0
Structure Layers Selection Groups Views		
Options - Pull		
@ General		
Properties	9 100mri 75.37mm	
	Bottom edge	
Properties Appearance	▲ Design * ×	4 Þ ×
Create round from 1 edge		🛛 -

Figure 9: Roundings.

- Repeat this rounding operation for top and bottom edges.

- Select the *Sketch mode* icon and select the bottom plane of Ahmed's body (Figure 10), then click *Plan View*.

Figure 10: Sketch plane definition.

- Use *Construction line* and *Offset Curve* to draw the construction geometry of Figure 11 that will be required to define the supporting feet of Ahmed's body:

File Design Insert Detail Display Measure Repair Prepare SheetMetal KeyShot	
Image: Control of the control of th	5YS R18.0
Structure Layers Selection Groups Views	
Options-Selection 9	
Li Stath O ^	
C Maintain skatch connectivy	
🕺 Skotch 💿	
Øsepe bord Øsepe banje Creete løpordarines → 372mm → 372mm	
Properties 0	
	d b x
it togen appendix a constraint and the Defendance a	

Figure 11: Bottom construction geometry.

- Using *Circle*, draw the base of cylindrical feet of Ahmed's body, specifying 30mm by keyboard as diameter (Figure 12):



Figure 12: Supporting feet geometry.

- Use *Pull* to extrude both circles with an extruding length of 50mm, defined by keyboard (Figure 13):



Figure 13: Supporting feet.

Now the geometry of the Ahmed's body is completed, but we need to define an external enclosure to simulate the external aerodynamics around the body. This enclosure starts 2L

before the body and extends 5L behind it; its height is 1.4m and its width is 2.5W (these dimensions are recommended for the 10th joint ERCOFTAC Workshop on Refined Turbulence Modelling).

- Click on *Prepare* tab and then click on *Enclosure* icon; select *Box* as Enclosure type and untick *Symmetric dimensions* (Figure 14):



Figure 14: Enclosure options.

- Spot V Weld S Volume Enclo ure 🤤 Imprin Orient ANSYS k. 🖣 \checkmark 6 • 1. + + • •
- Select the body and insert the dimensions reported in Figure 15, then click on \checkmark icon:

Figure 15: Enclosure dimensions.

- In the *Structure* tab, right click on *Solid* (the Ahmed's body) \rightarrow *Suppress for Physics* since we don't need to mesh it nor to perform calculations inside it (as it would in the case of a two-way fluid-structure coupling) (Figure 16):



Figure 16: Ahmed's body volume is not required for our CFD analysis.

Since we want to increase the mesh resolution around the body, we need to define a *Body of influence*: click on *Sketch* icon under *Design* tab and select the symmetry plane of the Ahmed's body (Figure 17):



Figure 17: Plane selection for Body of influence definition.

- Click on *Plan View* and define the rectangular geometry of Figure 18 (the x length of the body is 1044mm, the width of the rectangle is 2544mm):



Figure 18: Body of influence definition.

- In the *Structure* tab, untick *Solid* and *Enclosure* (Figure 19):

💐 늘 🖥 🤊 - ሮ - 🔹				A:Ahmed_body - FFF - SpaceClaim		- & ×
File Design Insert Detail	Display Measure	Repair Prepa	re Sheet Metal KeyShot			<u>~ 0</u>
Cilipboard Cilipboard Cilipboard Cilipboard	ヽ□ 0 ♪ 7 ♪ □ 0 ♪ 0 ∴ 0 0 ⑦ ↓ Sketch	1 × 0 ⊠ 1 × ⊡ @ 1 ≥ × Σ 0 More	de Edit	Sophit Body Sophit Body Sophit Body Sophit Body Sophit Sophit	Image: t Image: t Image: t	
Structure	4	Click to select. Do	puble-click to select a tangent cha	in. Double-click again to select a closed loop. Drag t	to edit your sketch.	
Solid Solid Solid Solid Construction line Construction line Construction line Construction line Construction line Structure Lyous Selection Groups View	•					ANSYS
Options - Selection	9.					
III Sketch	la 🔿 🔿					
Maintain sketch connectivity						
1 Sketch	8				i i	
Snap to grid Snap to angle Create layout curves				**		
III Dimensions	0					
Cartesian dimensions						
Polar dimensions	~					
		×+(€)			<i>b</i>) 8	
Properties Appearance		A FFF*×				4 Þ ×
Click to select. Double-click to select a tanger	nt chain. Double-click a	gain to select a closed l	loop. Drag to edit your sketch.	x=-2331	1,0125 y=455,5283 🛛 🌖 🔺	🕅 🕅 n 🖂 n 🕂 n 🕂 🔍 n 🛩 🥓

Figure 19: Body of influence definition.



- Click on *Pull* to extrude the rectangular profile in the +z direction for 400mm (Figure 20):

Figure 20: Body of influence.

- Rename the new solid as 'Size_box' (Figure 21):

2011 - 10 - 10 - 10 - 10 - 10 - 10 - 10	AvAhmed_body - FFF - SpaceClaim	- @ ×
File Design Insert Detail Display Measu	re Repair Prepare Sheet Metal KeyShot	∧ ⑦
Image: Second	1 1 X	
Structure P	Click an object. Double-click to select an edge loop. Triple-click to select a solid.	
		ANSYS R18.0
Structure Layers Selection Groups Views		
Options - Selection 0 Properties 0		
Properties Appearance	A FFF ×	4 Þ ×
Click an object. Double-click to select an edge loop. Triple-click to s	select a solid	Q 10 10

Figure 21: Ahmed's body enclosure.

- *File* \rightarrow *Save Project* and close SpaceClaim.

Now that every geometric entity has been defined, we can start meshing the enclosure using ANSYS Meshing.

2.2 Meshing the enclosure with ANSYS Meshing

In WB, right click on *Mesh* \rightarrow *Edit* to mesh the enclosure in ANSYS Meshing (Figure 22):

Ahmed_body - Workbench					-	_		×
File View Tools Units Extensions Jobs	Help							
🐑 😝 🖳 🕄								
Import • Reconnect	Opdate Project ACT Start Page							
Toolbox 👻 🕂 🗙 Pr	oject Schematic							→ 廿 X
 Eigenvalue Buckling Ejednvalue Buckling Electric Explicit Dynamics Fluid Flow - Blow Molding (Polyflo Fluid Flow (CEX) Fluid Flow (Fluent) Fluid Flow (Polyflow) Harmonic Acoustics Harmonic Response Hydrodynamic Diffraction Hydrodynamic Ciffraction Hydrodynamic Response IC Engine (Fluent) Modal Modal Modal Acoustics Random Vibration Response Spectrum Rigid Dynamics Component Systems Design Exploration ACT 	• A 1 C Fluid Flow (Fluent) 2 S Geometry 3 Mesh A 4 S Setup 5 Solution 6 6 Results Ahmed_body	Edit Duplicate Transfer Data From New Transfer Data To New Update Update Update Upstream Components Clear Generated Data Refresh Reset Rename Properties Quick Help Add Note	•					
Right-click to update component.		Job Monitor	👥 No Di	PS Connection	Show Progress	s 🏓	Show 1 Me	ssages

Figure 22: Starting a new Meshing session from WB.

- Under *Outline* tab \rightarrow *Project* \rightarrow *Model* \rightarrow *Geometry* right click on *FFF**Size_box* and select *Hide Body* (Figure 23):

M			Contex	t						A: A	hmed_	body - M	eshing [AN	ISYS AUTO	DDYN Pre	epPost]							- 8	×
File	2	Home	Geomet	ry	Display Selec	tion	Autor	nation													Quick Launch	1	^ 🗹	0-
Ob Gene	ject trator Took	Run Macro	Scripting Mechanica	N al U	lanage ser Buttons																			
Outlin	ne 👘		••••••••••••••••••	- 4	□× Q Q		e 🍣		- 🔶 🤇	20	0 0	Select	🐂 Mode	- 11		D D 🖲		y	Clipboard -	[Empty]	😜 Extend 🕶	Select By	·*	×
N	ame	*	Search Ou	utline	× ·																			
		del (A3) Geomet Geomet Geomet Materia Coordin Connec Mesh	try holosure/Enc FF/Size_box is nate Systems tions	osur ≇ ₽ ₽ ₽ ₽	Update Generate Mesh Preview Hide Body Hide All Other Bod Rename Group Suppress Body	lies F Ctrl+i	Hide	Body Hid sele bod	e cted ies.													ŗ	NSYS 2020 R1	Ì
Detail	ls of "FF	F\Size b	ox"	*	Suppress All Other	Rodies	0.																	
🗄 Gra	phics Pi	roperties			Create Named Sele	ection		ress F1 f	or nelp.															
- Def	inition	red	No	6	Transform Part																			
Co	ordinate	e System	Default Co	_	Export																		Y	
Tre	atment		None				-																4	
Ref	erence F	Frame	Lagrangiar		Update Selected Pa	arts	1																	
🖃 Ma	terial			٠	Clear Generated D	ata																	<u> </u>	
Ass	ignmen	t	Define d T		_								0,000	_		1,500		3	3,000 (m)			Z	х 🔪 х	¢ i
Flu	Id/Solid	Pau	Defined By	Geo	n									0,750)		2,250							
T BOI	inuing I	DUX			~																			
Hide	selecter	d bodies															🤨 1 Me	ssage	No Selection	 Metric () 	m. ka. N. s. V.	A) Degrees	rad/s Cel	Isius

Figure 23: Hiding the body of influence.

- Click the *Face* icon \bigcirc and select the inflow face (left click), then right click on it \rightarrow *Create Named Selection* and specify the name *Inflow* (Figure 24):



Figure 24: Named selection.

- Repeat this operation for all the faces of the enclosure (Top, Ground, Side, Symmetry, Outflow, Ahmed_surface). For the surface of the Ahmed's body: click on the z axis for a side view, click on *Select Mode* \rightarrow *Box Select* to easily select all its patches (Figure 25):



Figure 25: Ahmed's surface selection.

- Under *Outline* tab \rightarrow *Project* \rightarrow *Model* \rightarrow *Mesh*, in the *Details of "Mesh"* tab, specify all the marked properties as in Figure 26:

₩ 🔛 ₹	Context						A : Ahme	ed_body -	Meshing [ANSYS AUT	DDYN PI	epPost]							-	ъ×
File Home	Mesh	Display S	election	Automation														uick Launch		<u>∼ № 0</u> .
Object Generator Tools	Scripting Mechanical	Manage User Buttons																		
Outline					8.	00	6 6 6	0 D	0 - 4			Select 🐂	Mode-	57 m			n 🐺 💬	Clipboard -	[Empty	×
Name *	Search Outl				8	~ ~			• •	~~~	~								1	•
Project* Model (A3) Model (A3) Materials Coordina Coordina Materials Coordina Connection Mesh	y s ate Systems ions selections																		AN 20	SYS 020 R1
Details of "Mach"																				
Details of Wesh																				
Detauts		CED.		·	`															
Colver Preference		Eluant																		
Element Order		ipear																		
Element Size																				
Cising		2, 1 m																		
- Sizing		N.				п														
Ose Adaptive Sizing		NO				Л														
May Size		11 m				\mathbf{A}														
Mach Defeaturing		les l		_		v														
Defeature Size		1.e-003 m				🖃 Infla	tion				_									
Capture Cuprature		(ec				Use	Automatic	Inflation	Pr	ogram Con	trolled									Y
Cuprature Min Size		2 e-003 m				Infla	tion Optio	n	Fi	rst Aspect R	atio								- 1	
Curvature Normal	Angle	Default (18 %)			1	F	rst Aspect	Ratio	5,											•
Capture Proximity		(es				N	laximum La	yers	5										•	× 🔶
Provimity Min Size		5 e-003 m		<	7	G	rowth Rat	e	1,	2										
Num Cells Across	Gan	, - 303 m		N	/	View	Advanced	Options	N	0										
- Hum Cells Across	oup				1															
Ready													🔑 1 M	essage	No Select	ion 🔺 I	Metric (m. l	(a. N. s. V. A) [egrees ra	d/s Celsius

Figure 26: Mesh Sizing and Inflation properties.

- Since we want an extruded mesh with prisms in the boundary layers, we must define their locations; under *Outline* tab \rightarrow *Project* \rightarrow *Model* \rightarrow *Named Selections*, click on *Ground* and set *Include* under *Program Controlled Inflation* in the *Details of "Ground"* tab (Figure 27):

File ₹	Home	Contr Named Se	ext	Display	Selection	Autom	ation	A : Ahme	d_body - M	eshing [AN	SYS AUTOD	YN PrepPo	ost]					Ouick Launch	- 8 ^ (× 2 💁
■→ Object Generator Too	Run Macro	Scripting	Manage User Butt	ons																
Outline :					→ 		ତ୍ର ତ୍	1		- 💠 🔅	Q Q (() Sele	ct 🔩 M	ode - 😰 🕻) 🖪 🕅	🕅 🖷 🏹 🖻		[Empty]	×
Name	-	Search Out	line 🗸 .																	
	Coordina Connecti Mesh Named So	te Systems ons elections				^	Ground 24/04/202	10 14:17 nd											ANSY 2020	'S R1
	C Syn	e nmetry tflow und ned_surface				*														
Details of "G	iround" 📀				- 1	□ × □														
Scope																				
Scoping N	lethod	Ge	ometry Se	ection																
Geometry		1 F	Face																	
Definition																				
Send to So	olver	Yes	5																	
Protected		Pro	ogram Con	trolled																
Visible	optrolled	oflation Inc	slude			- 1														
Precenve D	uring Solve	(Beta) No	liuue			<u> </u>													Y	
- Statistics	Joing Joine	(0410) 110	•																	
Туре		Ma	anual																•	
Total S	election	1 6	ace																	• X
Surface	e Area	7,5	5157 m ²								C	0,000		0,200		0,40	00 (m)		-	
Suppresse	d	0											0 100		0 300					
Used by M	lesh Works	heet No)										0,.00		3,500					
														1 Messao	e No S	election	 Metric (n 	nka NsVA) D	eorees rad/s (Celsius

Figure 27: Inflation locations.

- Repeat the same operations for the *Ahmed_surface* Named Selection.



- *Outline* \rightarrow *Project* \rightarrow *Model*, right click on *Mesh* \rightarrow *Insert* \rightarrow *Sizing* (Figure 28):

Figure 28: Mesh sizings.

- Under *Outline* tab \rightarrow *Project* \rightarrow *Model* \rightarrow *Geometry*, right click on *FFF**Size_box* \rightarrow *Show Body*; click the *Single Select* icon \clubsuit Moder and *Body* icon B;

- Repeat the insertion of a new sizing: in *Details of "Sizing" - Sizing* tab, for *Scoping Method* select *Geometry Selection*, for *Geometry* select the whole enclosure, for *Type* select *Body of Influence*, for *Bodies of Influence* select the size box and for *Element Size* insert 0,030m (Figure 29).



Figure 29: Mesh sizings.



- Right click on $Mesh \rightarrow Generate Mesh$ to create the mesh (Figure 30):

Figure 30: Mesh detail around the body.

- Outline \rightarrow Project \rightarrow Model \rightarrow Geometry, click on Mesh to display the mesh; click on the top Mesh tab \rightarrow Insert \rightarrow Section Plane to insert a section plane for exploring the volume mesh (Figure 31). Click on Show Whole Elements icon \implies to display whole mesh elements and not their projection on the section plane:



Figure 31: Mesh prism layers around the body surface and the ground.

The meshing phase is now complete: the mesh is composed by 380k tetrahedrons (*Tet4*, 4 nodes) for the volume mesh and 100k triangular prisms (*Wed6*, 6 nodes) in the inflation layers near the walls.

- *File* \rightarrow *Save Project...* to save the mesh into the WB Project and close ANSYS Meshing.

2.3 Setup the problem in Fluent

In WB, right click on *Setup* \rightarrow *Edit* to launch Fluent (Figure 32):



Figure 32: Starting Fluent from WB.

- In the Fluent Laucher window, untick Double Precision Option (Figure 33):

F Fluent Launcher 2020 R1 (Setting Edit Only)	- 0	×
Fluent Launcher	ANS	SYS
Simulate a wide range of industrial applications us general-purpose setup, solve, and post-processing of ANSYS Fluent	sing the g capabilit	ies
Dimension		
0 25		
(•) 3D		
Options		
Double Precision		
Display Mesh After Readition	ina	
\bigcirc Po pot show this popula	agin	
	iyani	
Load ACT		
Parallel (Local Machine)		
Solver Processes	1	\$
Solver GPGPUs per Machine	0	\$
✓ Show More Options ✓ Show Learning Reso	ources	
Start Cancel Help	•	

Figure 33: Fluent settings.

- Under the *Tree* tab \rightarrow *Setup* \rightarrow *General*, click on *Check* and *Report Quality* to check the mesh quality and set the marked settings as in Figure 34:



Figure 34: Fluent general Settings.

- Under Setup \rightarrow Models double click on Viscous - SST k-omega to specify a Realizable k- ε turbulence model with Non-Equilibrium Wall Functions as in Figure 35:



Figure 35: Turbulence model.

- Under Setup \rightarrow Materials \rightarrow Fluid, click on air to check the correct values of Density and (Dynamic) Viscosity as in Figure 36:



Figure 36: Air properties.

- Setup \rightarrow Cell Zone Conditions, double click on *enclosure_enclosure* to check our enclosure is filled with *Fluid* air as in Figure 37:



Figure 37: Cell zone conditions.

- Under Setup \rightarrow Boundary Conditions, right click on the chosen Named Selection \rightarrow Type to set the correct boundary condition (Figure 38): side, symmetry and top boundary conditions must be set to symmetry type (no shear stress), ahmed_surface and ground boundary conditions must be set to wall;



Figure 38: Boundary condition types.

- For *inflow*, set it as *velocity-inlet* type with the marked settings of Figure 39:



Figure 39: Inflow conditions.

- For *outflow*, set it as *pressure-outlet* type with the marked settings of Figure 40:



Figure 40: Outflow conditions.

- Under Setup \rightarrow Reference Values, specify inflow under Compute from and $A_x = 0.057516$ m² as Area (Figure 41):





- Under Solution \rightarrow Methods select the Coupled method for the Pressure-Velocity coupling and First Order Upwind (for now) for Momentum, Turbulent Kinetic Energy and Turbulent Dissipation Rate (Figure 42):



Figure 42: Solution settings.

- Under Solution \rightarrow Controls specify 0.3 for both Momentum and Pressure Pseudo Transient Explicit Relaxation Factors (Figure 43):



Figure 43: Solution controls.

- Under *Solution* \rightarrow double click on *Report Definitions* to define new quantities to be reported; select *New* \rightarrow *Force Report* \rightarrow *Drag* to define a drag report (Figure 44):



Figure 44: Drag report definition.

- Under *Drag Report Definition* specify the settings of Figure 45 to define a drag coefficient report:

A:Ahmed_b	ody Parall	el Fluent@E	ESKTOP-TG6RB5	3d, pbns	rke] [ANSY	S CFD Enter	prise]										a ×
9 😫 🖄 H	⇒ ∧ 1	3 H2														_	_
File	Domain	PI	iysics Usei	-Defined	Solutio	in	Results	View	Parallel	Design 🔺					Q Quick Search (Ctrl+I	•) •	E ANSYS
		Mesh	_			Zones	_	Interfaces	Mesh Models	Turbo Model	Adapt	Surface					
Display			Scale	😂 Com	bine 🕌	Delete	Append	- Mesh	🔗 Dynamic Mesh	Enable	Refine / Coar	sen + Create	-				
i Info 🖕		1999 C	📣 Transform	💡 🖵 Sepa	rate 💡 🖽	Deactivate	Replace Me	sh I Overset	C Mixing Planes	Turbo Topology		🔏 Manage	e				
🖌 Units	Check-	Quality 🗸	🔶 Make Polyhedra	🛛 ∻ Adja	cency 📑	Activate	唱 Replace Zor	1e		Turbo Create	ooo More						
Outline View			٢	Task Page			< [Mesh					×
Filter Text				Solution C	ontrols		?			🖪 Drag Report Defin	ition			×	< <u> </u>		
Setup				Pseudo Tr	ansient Exp	licit Relaxat	ion Factors			Name	1				\leq		
General				Pressure			Q			cd	J						
Ø Models				0.3						Options	Re	port Output Type					
	• e Condition			Momentur	1						L.	Drag Coefficient					
Boundar Boundar	y Condition	5		Report I	Definitions				×			Drag Force			<		
🕫 Mesh In	terfaces									Per Zone		all Transform					
🗿 Dynamii	Mesh			Denvel Defe	Norma [0/0]		Report Defin	ition Properties		Average Over(Iterations)	Ē	al zones (riller real	(
🖄 Referen	e Values			Report Denn	none (ovo)	<u>v</u> .	Name		-	1 0	L.	hmed_surface					
K. Reteren	e Frames						Field			Force Vector	9	round					
Solution	copressions						Surface/Zone	Names :		Y Y Z							
% Method	5						Per Surface/2	ione :		1 0 0							
× Controls							Average Over	r a									
📧 Report E	efinitions			Nous	Delete	Compute				Report Files [0/0]	= 5 5						
Q Monitor	5			New JE	Delete	compute									\sim		
E Initializa	tion			Create Out	put Paramel	ter											
	on Activitie	5															
Run Cali	ulation			Used In						Report Plots [0/0]							X.
 Results 							Report Fil	e Definitions					11/21				
 Surfaces Graphic 							Report Plo	t Definitions					10/41			z	
I Plats																	10
Animation	ons									Create							
💿 🖻 Reports				Edit						✓ Report File							 0 selected
 Parameters 	& Custom	zation	-							C Report Plot							@ <
							Close Help			Frequency 1 🤤							A
								maximum face are	(m2): 9,485917e	Print to Console							
							CI	hecking mesh		Create Output Paramet	er 🗆	Highlight Zones					
							Me	sh Quality:					-				
											ОК Соп	npute Cancel H	lelp				
							Mit	nimum Orthogonal o improve Orthogo	Quality = 1.0335 mal quality , use	"Inverse Orthogonal	Quality" in Fl	uent Meshing.			p1 4.11292e-02	1.42019e-01)	
							set	here Inverse Orth	ogonal Quality =	1 - Orthogonal Qual:	τ¥)						
							Max	ximum Aspect Rati	o = 2.19214e+01	cell 12348 on zone :	(ID: 12349 on	partition: 0) a	t location (-	5.51506e-01 5.	29889e-02 1.9484	9e-01)	-
																	×

Figure 45: Drag report definition.

- Repeat the previous procedure to define a Lift Report Definition if needed.

- Under Solution \rightarrow Monitor \rightarrow Residual tick Show Advanced Options and set the Convergence Criterion to none since we'll define a fixed number of iterations (Figure 46):



Figure 46: Residual monitors settings.

- Under Solution \rightarrow Initialization select Hybrid Initialization and click Initialize (Figure 47):



Figure 47: Solution initialization.

- Under Solution \rightarrow Run Calculation set 50 as Number of Iterations and click Calculate (Figure 48); anyway, we can stop the calculation before the 50 iterations if we believe the solution to be acceptable (checking residuals or report plots, for example).



Figure 48: Residual history plot.

- When the solution "converged" (we are employing first order upwind schemes), we can go back to *Solution* \rightarrow *methods* to select *Second Order Upwind* for Momentum, Turbulent Kinetic Energy and Turbulent Dissipation Rate (Figure 49) to improve the calculations (less numerical viscosity dissipation); run the simulation for 200 additional iterations for example.

A:Ahmed_body Par	allel Fluent@	DESKTOP-TG6RB59	3d, pbns, rke]	[ANSYS CFD Ente	rprise]												- 0	×
Ø & @ ∂ ≠ Λ □ ≅																		
Eile Domai	in F	hysics User	-Defined	Solution	Results	View		Parallel D	esign 🔺						Q Quick Search	(Ctrl+F)	• E	ANSYS
Display Display The Check	Mesh (* Quality -	Scale Transform Make Polyhedra	© Combine □ Separate ∲ Adjacency	Zones - Delete - Deactivate / Activate	Append Beplace	Mesh	erfaces Mesh Dverset	Mesh Models	Turbo Mode Enable	el	Adapt Refine / Coarsen More	Surface + Create & Manage						
Outline View		<	Task Page		<	o 🚨		Scaled Residuals		× 🗖		cd-rplot		× 🗖		cl-rplot		×
Filter Text			Solution Metho	ods	(?)	*	Residu	lais										
Setup G. General © Models © Models © All Zone Condition © All Zone Condition Ø Models © Gell Zone Condition Ø Models Ø Models © Gell Zone Condition Ø Models Reference Values Named Expression Report Definition © Gentral © Kontors Besidual © Report Fless © Caliston Activ © Runc Liculation © Surfaces © Surfaces	ions ions is sons 15 Conditions titles		Pressure -Veloc Scheme Coupled Spatial Discret Gradent Least Squares Pressure Second Order Momentum Second Order Turbident Kinet Second Order Turbident Disg Second Order Turbident Disg Second Order Pressure Pressure Pressure Pressure Hard Pressure Hard Pressure Hard Pressure Hard Pressure Hard Pressure Hard Pressure Hard Pressure Second Order Turbident Disg Second Order Turbident Disg Second Order Second Order Second Order Second Order Second Order Second Order Second Order Hard Pressure Hard Pressure Hard Pressure Hard Pressure Hard Pressure Hard Pressure Pressur	ization Cell Based Cell Based Cell Based Upwind Cell Energy Upwind Cell Cenergy Upwind Cell Cenergy Cell Cenergy Cene	•		-velocity -velocity -velocity -velocity psilon		1e-00 1e-01 1e-02 1e-03 1e-04 1e-05 1e-05 1e-06 1e-06 0		50	100	150 Iterations	20	0	, 250	300	
a Scene						all												• 0 selected
Animations Animations						Canaala												
 Parameters & Custo 	mization					hyb_ini hyb_ini Done. Calculation	t-0 t-1 complete											*

Figure 49: Switching to Second Order Upwind.

The history plot of drag coefficient C_D is reported in Figure 50; we can see that with First Order Upwind scheme (=till 50 iterations) for all the transported quantities (velocities and turbulent quantities), C_D "converged" to 0.45, while the Second Order Upwind scheme (=above 200 iterations) decreased C_D to 0.334. This is in perfect agreement with the upwind schemes: first order scheme is more diffusive than second order and simulates a higher viscosity flow with higher drag coefficients.



Figure 50: Drag & lift coefficients history plot.

The comparison of the computed drag coefficient to other experimental and numerical results is reported in Figure 51; as we can see our result is in good agreement with both experimental and numerical data from literature, even if it is computed through a RANS simulation with a not so refined mesh.



Figure 51: Computed drag coefficient vs other results (from [2]).

- *File* \rightarrow *Save Project* and close Fluent.

2.4 Visualization of results in CFD-Post

In WB, right click on *Results* \rightarrow *Edit* to launch CFD-Post (Figure 52):



Figure 52: Starting CFD-Post from WB.

- In the *Outline* tab, under *Cases* \rightarrow *Ahmed_body* \rightarrow *enclosure-enclosure*, select only the *ahmed_surface* and the *ground* components. Click on the *Location* icon \bigcirc and select *Line*, then specify the marked settings as in Figure 53 \rightarrow *Apply*:



Figure 53: Location definition.

- Click on the *Streamline* icon \cong and specify the marked settings as in Figure 54 \rightarrow *Apply*, to view the streamlines in the symmetry plane in the rear separation zone:



Figure 54: Streamline definition.

A comparison between computed and experimental (mean) streamlines along the symmetry plane in the rear separation zone is reported in Figure 55: although the relative position of recirculation bubbles is not exactly computed, we can see that an approximate width of 200mm is computed with good agreement to the experimental width.



Figure 55: Comparison of streamlines in the rear separation zone (from [4]).

A comparison between computed and experimental (mean) streamwise velocity profiles along the symmetry plane in the rear separation zone is reported in Figure 56: the computed profiles show a discrete agreement with experimental ones.



Figure 56: Comparison of streamwise velocity profiles in the rear separation zone (from [3]).

CFD-Post is a powerful tool to analyze and display the results of a CFD simulation in a graphical way; be careful to choose a meaningful way to present your results (see Figure 57).



Figure 57: Suggestive CFD image with CFD-Post.

References

- [1] S.R. Ahmed, G. Ramm, and G. Faltin. Some salient features of the time-averaged ground vehicle wake. In *SAE Technical Paper*. SAE International, February 1984.
- [2] F.J. Bello, T. Makela, L. Parras, C. del Pino, and C. Ferrera. Experimental study on ahmed's body drag coefficient for different yaw angles. *Journal of Wind Engineering and Industrial Aerodynamics*, 157:140 – 144, 2016.
- [3] C. Hinterberger, M. Garcia-Villalba, and W. Rodi. Large eddy simulation of flow around the Ahmed body. In R. McCallen, F. Browand, and J. Ross, editors, *Lecture Notes in Applied and Computational Mechanics/The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains.* Springer, New York, 2004.
- [4] H. Lienhart, C. Stoots, and S. Becker. *Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model)*, pages 323–330. Springer Berlin Heidelberg, Berlin, Heidelberg, 2002.