Riccardo Zamolo, Enrico Nobile

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

Esercitazioni di Termofluidodinamica Computazionale

2D differentially heated cavity with ANSYS Fluent and MATLAB



April 2023

1 Introduction

1.1 Problem definition

This tutorial document shows how to perform a CFD analysis of a 2D differentially heated cavity using ANSYS Workbench 2022 (hereafter "WB") for the simulations and MATLAB for some postprocessing calculations.

The differentially heated cavity is a typical CFD benchmark problem which involves a buoyancy-driven flow due to density variations, which are in turn due to temperature variations, i.e., natural convection:

"Buoyancy-driven flow in a square cavity with vertical sides which are differentially heated is a suitable vehicle for testing and validating computer codes used for a wide variety of practical problems." [1]

With reference to Figure 1, a fluid is enclosed in a 2D square cavity with side length L where the left and right walls are kept at fixed temperatures T_h and T_c , respectively, while the horizontal walls are adiabatic, i.e., no heat transfer. The fluid is subjected to the gravity g along the vertical direction y; in other words the 2D cavity can be imagined in a vertical plane: this is necessary for the appearance of a buoyancy-driven flow due to density variations.

Intuitively, for a given fluid, the intensity of the buoyancy forces increases as both the temperature difference $\Delta T = T_h - T_c$ and the dimension L increase. Therefore, weak convective motions are expected for small cavities with small ΔT , while stronger convective flows are expected for big cavities with large ΔT . Accordingly, it can also be expected that sufficiently strong buoyancy forces will lead to unsteady flows, while steady flows are expected for weak buoyancy forces. In this document we will focus on the latter case, i.e., a laminar, steady-state natural convection problem.

1.2 Governing equations

The aformentioned steady-state problem is described by the following coupled conservation equations of mass, momentum and energy with Boussinesq approximation, i.e., linear depen-



Figure 1: 2D differentially heated cavity: schematic representation of the problem (left) and example of temperature field for $Ra = 10^5$ and Pr = 0.71 (right).

dence of density on temperature in the buoyancy term alone:

$$\nabla \cdot \boldsymbol{u} = 0 \tag{1}$$

$$(\boldsymbol{u}\cdot\nabla)\boldsymbol{u} = -\frac{1}{\rho_0}\nabla p + \nu\nabla^2\boldsymbol{u} + g\beta(T-T_0)\boldsymbol{e}_y$$
(2)

$$\boldsymbol{u} \cdot \nabla T = \alpha \nabla^2 T \tag{3}$$

where e_y is the unit vector along y, ρ_0 is the reference density, respectively, ν is the kinematic viscosity, g is the gravitational acceleration, β is the thermal expansion coefficient and α is the thermal diffusivity. T_0 is the reference temperature, which can be assumed to be the mean temperature $(T_h + T_c)/2$. Other choices for the reference values of temperature and density in equation (2) are equivalent from a mathematical point of view since they only affect the pressure by a multiplicative constant ρ_0 and by a static head $g\beta T_0\rho_0$.

The physical parameters of the problem are 6: L from the geometry, ΔT from the boundary conditions, ρ_0 , ν , $g\beta$ and α from equations (1)-(3). The involved fundamental units are 4: length, mass, time and temperature. The Buckingham theorem (BT) states that the problem is then completely described by 6 - 4 = 2 adimensional groups (the dimensional matrix has rank 4). For example, the following groups can be obtained (they are not unique):

$$\pi_2 = \frac{g\rho\Delta TL}{\nu\alpha} =: \text{Ra} \quad (\text{Rayleigh number}) \tag{5}$$

Pr is the ratio between the momentum and thermal diffusion coefficients, i.e., kinematic viscosity ν and thermal diffusivity α , and therefore it is strictly related to the relative thickness of the momentum and thermal boundary layers. It depends only on the fluid: Pr \ll 1 for liquid metals, Pr \approx 1 for gases, Pr \gg 1 for oils.

Ra is the ratio between the characteristic time scales of diffusive and convective thermal transport due to conduction and buoyancy, respectively. Therefore, the larger the Ra, the faster the convective dynamics compared to the conductive one. For a given Pr, the transition from steady to unsteady flows is controlled by Ra only. In this document the chosen fluid is air with Pr = 0.71, for which the onset of unsteadiness occurs for Ra $\approx 1.82 \times 10^8$ [3].

Since the ratio between two adimensional groups is adimensional as well, another choice for the adimensionalization could be obtained by considering Ra/Pr instead of Ra:

$$\frac{\text{Ra}}{\text{Pr}} = \frac{g\beta\Delta TL^3}{\nu^2} =: \text{Gr} \quad \text{(Grashof number)}$$
(6)

which is the ratio between buoyancy and viscous forces: the larger the Gr, the stronger the convection due to buoyancy.

Without manipulating equations (1)-(3), the dimensional analysis tells also us how to scale the dependent variables of the problem, i.e., nondimensionalization: trivially, the position \boldsymbol{x} and the temperature difference $T - T_0$ can be scaled by L and ΔT , respectively. The dimensional analysis then states that the velocity \boldsymbol{u} and the pressure p can be scaled by $u_0 = \alpha/L$ and $\rho_0 u_0^2$, respectively (again, these choices are not unique). The same values can be obtained by a direct nondimensionalization of equations (1)-(3). By considering the following scalings:

$$\hat{\boldsymbol{x}} := \frac{\boldsymbol{x}}{L}, \quad \hat{\boldsymbol{u}} := \frac{\boldsymbol{u}}{u_0}, \quad \hat{T} := \frac{T - T_0}{\Delta T}, \tag{7}$$

equation (3) takes the following form:

$$\hat{\boldsymbol{u}} \cdot \hat{\nabla} \hat{T} = \frac{\alpha}{u_0 L} \hat{\nabla}^2 \hat{T} \tag{8}$$

where the coefficient $\alpha/(u_0L)$ must be nondimensional, since every other term is nondimensional as well. We can therefore choose $\alpha/(u_0L) = 1$ for the sake of convenience, i.e., $u_0 = \alpha/L$, as previously obtained.

Equation (2), on the other hand, becomes:

$$(\hat{\boldsymbol{u}}\cdot\hat{\nabla})\hat{\boldsymbol{u}} = -\frac{1}{\rho_0 u_0^2}\hat{\nabla}p + \frac{\nu}{\alpha}\hat{\nabla}^2\hat{\boldsymbol{u}} + \frac{g\beta\Delta TL^3}{\alpha^2}\hat{T}\boldsymbol{e}_{\hat{y}}$$
(9)

and again, since every term must be nondimensional, the following nondimensional terms arise, as previously obtained:

$$\frac{p}{\rho_0 u_0^2} =: \hat{p}, \quad \frac{\nu}{\alpha} = \Pr, \quad \frac{g\beta\Delta TL^3}{\alpha^2} = \frac{g\beta\Delta TL^3}{\nu\alpha} \frac{\nu}{\alpha} = \operatorname{Ra}\Pr$$
(10)

In the end, the nondimensional form of equations (1)-(3) is therefore:

$$\hat{\nabla} \cdot \hat{\boldsymbol{u}} = 0 \tag{11}$$

$$(\hat{\boldsymbol{u}}\cdot\hat{\nabla})\boldsymbol{u} = -\hat{\nabla}\hat{p} + \Pr\hat{\nabla}^{2}\hat{\boldsymbol{u}} + \operatorname{Ra}\Pr\hat{T}\boldsymbol{e}_{\hat{y}}$$
(12)

$$\hat{\boldsymbol{u}} \cdot \hat{\nabla} \hat{T} = \hat{\nabla}^2 \hat{T} \tag{13}$$

which depends only on the two adimensional numbers Ra and Pr as previously obtained from the BT, i.e., \hat{u} , \hat{p} , $\hat{T} = f(\text{Ra}, \text{Pr})$.

By considering the heat transfer coefficient h as another physical (dependent) variable defined by the wall heat flux $q = h\Delta T = -k\partial T/\partial x$, where k is the thermal conductivity of the fluid, we can also obtain the following adimensional group:

$$\pi_3 = \frac{hL}{k} =: \operatorname{Nu} \stackrel{\text{BT}}{=} f(\operatorname{Ra}, \operatorname{Pr}) \quad (\text{Nusselt number})$$
(14)

Nu is therefore the ratio between the heat flux at the boundary and the reference one due to ΔT , i.e., $k\Delta T/L$. Equation (14) can be interpreted both locally, i.e., at a given height y on the isothermal walls, and globally, i.e., by considering the integral mean over each isothermal wall:

$$\operatorname{Nu}_{y} = \frac{-\frac{\partial T}{\partial x}\Big|_{x=0,L}}{\frac{\Delta T}{L}} = -\frac{\partial \hat{T}}{\partial \hat{x}}\Big|_{\hat{x}=0,1}$$
(15)

$$\overline{\mathrm{Nu}} = \frac{1}{L} \int_0^L \mathrm{Nu}_y \, dy = -\int_0^1 \frac{\partial \hat{T}}{\partial \hat{x}} \Big|_{\hat{x}=0,1} \, d\hat{y} \tag{16}$$

which are thus, by their own definition, functions of Ra and Pr only, as stated by the BT.

2 Workbench project

The geometry of the square cavity will be defined using SpaceClaim, the uniform cartesian mesh will be generated using ANSYS Meshing and the problem will be solved using ANSYS Fluent. Each of these components is directly available from WB as shown in Figure 2.

- Start WB, *Toolbox* tab on the left \rightarrow drag&drop a *Fluid Flow (Fluent)* component to the main white window (*Project Schematic*), rename it as *Heated cavity*

- Right click on Geometry \rightarrow Properties (menu on the right) \rightarrow Advanced Geometry Options
- \rightarrow Analysis Type \rightarrow 2D
- *File* \rightarrow *Save as...* \rightarrow project name: <u>HC</u>
- Right click on *Geometry* \rightarrow *New SpaceClaim Geometry*... to start SpaceClaim:

🌉 HC - Workbench					_		×
File View Tools Units Extensions Jo	obs Help						
🖺 💕 🛃 🔣 🗍 Project							
Timport	t 🥖 Lindate Project 📲 ACT Start Page						
Toolbox • # X Proj	ect Schematic		т д х	Propertie	es of Schematic A2: Geometry		↓ ↓ ×
Analysis Systems					A	в	^
G Coupled Field Harmonic				1	Property	Value	2
😣 Coupled Field Modal	▼ A			12	Line Bodies		_
Coupled Field Static	1 🕄 Fluid Flow (Fluent)			12	Baramotore	Index	
Coupled Field Transient	2 Geometry			15	Parameter Kov	ANCIDE	-
Eigenvalue Buckling	3 📦 Mesh 🔹 🚺	New SpaceClaim Geometry		15	Attributes	AN3,03	_
Electric Explicit Dynamics	4 🚵 Setun	New DesignModeler Geometry		15	Named Selections		
Eluid Elow - Blow Molding (Po	r Galin G	New Discovery Geometry		17	Material Properties		
Fluid Flow - Extrusion (Polyflo	5 Solution	Import Geometry		18	Advanced Geometry Ontions		- N
S Fluid Flow (CFX)	6 💓 Results 👕		_	10	Analysis Type	20	
S Fluid Flow (Fluent with Fluent	Heated cavity	Duplicate		20	Lise Associativity		
🖾 Fluid Flow (Fluent)		Transfer Data From New	•	-20	Import Coordinate		$\left \right $
S Fluid Flow (Polyflow)		Transfer Data To New	•	21	Systems	5	ا م
Harmonic Acoustics		Lindate		22	Import Work Points		
M I S-DYNA		Lindate Linstream Components		23	Reader Mode Saves		
LS-DYNA Restart	1	Bafrach			Updated File		_
Magnetostatic	142			24	Import Using Instances	V	
1 Modal		Reset		25	Smart CAD Update	V	
Modal Acoustics	ab	Rename		26	Compare Parts On Update	No	•
🔟 Random Vibration		Properties			Enclosure and	_	_
W Response Spectrum		Quick Help		27	Symmetry Processing	v	
Rigid Dynamics		Add Note		28	Decompose Disjoint Geometry	V	-
Double-click component to edit.		Ich Monitor	No DP	S Connect		ow 0 Mess	anec
. ·		SOD MONITOR		o comicu		011 0 11033	- see

Figure 2: WB project (Fluid Flow with Fluent, 2D), starting SpaceClaim.

Regarding the physical parameters to consider in our numerical simulations, we have seen in Section 1.2 that any combination of L, ΔT , ρ_0 , ν , $g\beta$ and α is equivalent as long as the Ra and Pr numbers are the same. Therefore, for the sake of simplicity, it is easier to refer to the nondimensional equations (11)-(13) which depends only on Ra and Pr, and which are based on a cavity with unit side length L and unit temperature difference ΔT (see equations (7)). By comparing the coefficients of the nondimensional equations (11)-(13) with the ones of the dimensional equations (1)-(3) we can easily see that they match when, for example:

-
$$\rho_0 = 1 \ kg/m^3$$

-
$$\alpha = k/(\rho_0 \cdot c_p) = 1 \ m^2/s$$
, i.e., $k = 1 \ W/(m \cdot K)$ and $c_p = 1 \ J/(kg \cdot K)$

- ν is numerically equal to \Pr in $kg/(m\!\cdot\!s)$ units
- g is numerically equal to Ra in m/s^2 units
- β is numerically equal to Pr in K^{-1} units

which are the values employed in the following (with L = 1 m and $\Delta T = T_h - T_c = 1 K$).

2.1 Geometry definition with SpaceClaim

- Click on *Sketch mode* icon \mathbb{Z} at the bottom of the main graphical window and move the mouse position the sketch on the x - y plane (or click on the z axis)

- Click on *Plan view* icon B Plan View to have an orthogonal plan view:



Figure 3: Sketch on the x-y plane.

- Using *Rectangle* (or *Line*), define a square with side length L = 1 m = 1000 mm with the bottom left corner at the origin of the x - y axes:



Figure 4: Sketch of the 2D square cavity.



- Click on *End Sketch Editing*:

Figure 5: 2D square cavity.

- *Structure* tab on the left \rightarrow right click on *Surface* \rightarrow rename it to *Cavity*:





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

6

2.2 Meshing the cavity with ANSYS Meshing

Now that the cavity has been defined, we can start meshing it using ANSYS Meshing. Since the geometry is a square, we will create a simple structured cartesian mesh. - In WB, right click on *Mesh* \rightarrow *Edit* to start ANSYS Meshing:

🛃 HC - Workbench					_		×
File View Tools Units Extensions Jobs Help							
Constant Active Project Vupdate Project Active Active Active Project Project Schematic	age		- I - Y	Propertie	es of Schematic A3: Mesh		- 1 -
			* 4 ^	Troper of	a of Schemole AS. Mean		* 4 *
Constant Systems				<u> </u>	A	8	
Coupled Field Madal				1	Property	Valu	e
Coupled Field Static 1 C Eluid Flow (Fluent)				2	 General 		
Coupled Field Transient				3	Component ID	Mesh	
Elgenvalue Buckling	<u> </u>			4	Directory Name	HH	
Electric	Ť=	r. 19.		5	 Notes 		
🐱 Explicit Dynamics 4 🙀 Setup		Edit		6	Notes		
S Fluid Flow - Blow Molding (Po 5 🕼 Solution	7	Duplicate		7	Used Licenses	_	
G Fluid Flow - Extrusion (Polyflo	-	Transfer Data From New	•	8	Last Update Used		
G Fluid Flow (CFX)	8	Transfer Data To New			Econocia		
S Fluid Flow (Fluent with Fluent Heated cavity			<u> </u>	9	Dhueice	CED	
Fluid Flow (Fluent)	1	Update		10	Physics	Anu	
Fiuld Flow (Polyflow)		Update Upstream Components		11	Analysis	Acty	
Harmonic Acoustics		Clear Generated Data		12	Solver	FLUEINI	
	ক	Refresh		13	■ Mesh		
I S-DYNA Restart	_	Reset		14	Save Mesh Data In Separate File		
Magnetostatic	ab	Rename					
1 Modal		Descention					
B Modal Acoustics		Properues	_				
III Random Vibration		Quick Help					
100 Response Spectrum		Add Note					
Rigid Dynamics	_						
Static Acoustics							
Static Structural *							
View All / Customize							
Ø Double-dick component to edit.		ملام Job Monito	r 💷 N	o DPS Conr	nection I Show Progress	😃 Show 0 Me	essages .:

Figure 7: Starting ANSYS Meshing from WB.

- Click on the *Edge* icon $\boxed{10}$ \rightarrow click on the left wall \rightarrow right click on it \rightarrow *Create Named Selection* and specify the name *Hot*:

File Home M	ntext fesh Display	Selection Auto	mation			A : 1	leated cavity - Mes	hing [CFD Pr	epPost - CFD Ba	se]					Quick Laun	ch	- 8 X	K 0-
Duplicate Q Outline	Update Generate Mesh	Surface Source/Targ Mesh Preview	Method Si	zing Face Meshing	Mapped Meshing	Mesh Copy Match Contro	Contact Sizin	g 💷 Inflatio Mesh Gro 🗲 Weld	un 萨 Connect	Contact Ma	Froup	Mesh Me Meth Numb Mes	the Manual Mer Shering Connection Sh Edit	in Contact Nod Match Merg	e Node Move	Metric Graph *	Iges Display	2
Outline				• 4 ⊡ ×	Q	ର୍ 📦 🗑	8 🖻 🔶 - 🤞	. Q Q @) @ Select	🗙 Moder 😰 🗈	B B		¥ 👎 🖻 📋	Clipboard + [E	mpty] 🧯	Extend - 🧕	Select By -	•
Name v Sea Project* → Model (A3) B → Model	nports Systems							5	Selection Name	2		×				/	2022 F	5 21
								•	Apply selected Apply geometry Size Type Location X Location Y	geometry / Items of same:								
Details of "Mesh"				- 4 ⊡ ×					Location 2			- 64						
Display Display	Ura Caomatri	Satting		· · · · · ·						mannonding Mark No	dae							
- Defaults	ore dedinetry	secony								in caper any preserve		-						
Physics Preference	CFD									or l c								
Solver Preference	Fluent									UK Land	:el							
Element Order	Linear																	
Element Size	Default (7,071	1e-002 m)					_										1	
Export Format	Standard																1	
Export Preview Surface M	lesh No																•	
+ sizing																	ن	х
+ Quality									0,00	0	0,250		0,500 (m)					
Batch Connections																		
D to The Conflections				`						0,125		0,375			_			

Figure 8: Named selection.

- Repeat this operation for each wall of the cavity (*Cold* for the right wall, *Top*, *Bottom*).

Construction of the construction of th																					
Version Note: <	🚰 🖽 ₹	C	ontext						A:H	eated cavity - Mes	hing [CFD PrepP	ost - CFD Bas	e]								- 8 ×
Image: Second	File Home		Aesh Display Selectio	n	Automation														Quick Lau	inch	^ 🗹 📀
Outline •••••••••••••••••••••••••••••	Duplicate Q In Outline Ir	isert	Update Generate Mesh	Source	/Target Method	Sizing Face Meshing	Mapped Meshing	Mesh Copy	Match Control Control	Contact Sizin	Mesh Group	🗲 Connect	Contact Mate	ction Group th Group Group	Mesh Edit	Mesh Numbering Mesh Edi	Manual Mest Connection	Contact N Match M	ode Node	Metric Ed Graph *	Iges Max
Secto Outlow Former Former Station Former	Outline						i o	0) 	0000	Select	K Moder 🖅 🕞	1 G G	m m	h 🕅 🏋	🤋 🖻 🛅 c	lipboard +	Empty]	Extend -	Select By * 🚆
Prest*i Control Yaports Control Yaports Substrate Substrate Substrate Substra	Name	▼ Sei	arch Outline 🗸 🗸																		•
Instit Noth Method Image: Specific Method Specific Method Specific Method Image: Specific Method Method Specific Method Image: Specific Method Method Specific Method Image: Specific Method Method Method <tr< td=""><td>Project* Project* Model (A: </td><td>3) ietry In ietry rials inate</td><td>nports Systems</td><td>_</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>/</td><td>2022 R1</td></tr<>	Project* Project* Model (A:	3) ietry In ietry rials inate	nports Systems	_																/	2022 R1
Build of "Mech" Image State Petallo of Thematic State Image State Image State Image State Imate		1	Insert 🕨	\$	Method																
Generate Mech Review Revealer Mech Revealer Rev		2	Update	ផ្ទ	Sizing																
Preview Face Meshing Show Wath Corts © Create Finance The Show © Display Shoke Weind Concretion © Display Shoke Weind Concretion © Display Shoke Weind Concretion © Display Shoke Node Marge Group © Machine Node Marge Group Ensement Conder Linean Ensement Standard Node Marge Encord Format Sandard Encord F		ş	Generate Mesh	A	Refinement																
Bow Meth Corput Group All Suncivation Math Controls Group All Suncivation Path Group All Suncivation Prest FI for help. Details of "Meth" The Reventor Details of "Meth" Meth Controls Display Prest FI for help. Display Splot Use Generated Data Display Splot Weth Controls Display Splot Node More Display Splot Node More Display Topic State Meth Node More			Preview >	0	Face Meshing																
			Show	96	Mesh Copy																
Croup Al Similar Childrer Produ Product of Standard Tacs. Petrik of Mech* Product of Standard Tacs. Press F1 for help. Details of Mech* Weid Standard Tacs. Press F1 for help. Details of Mech* Weid Standard Tacs. Press F1 for help. Details of Mech* Weid Standard Tacs. Press F1 for help. Display Weid Standard Tacs. Press F1 for help. Display Node Merge Grapp Mech Edit Node Merge Grapp Display for Display Node Merge Grapp Node Merge Grapp Node Merge Grapp Display for Display		ş	Create Pinch Controls		Match Control	Face Meshing															
Cear Greenter Data Cear Greenter Data Cear Greenter Data Stat Becording Details of "Meth" Display Style Details of "Meth" Display Style Display Style		-	Group All Similar Children	0	Pinch	Ena	ble the g	eneration													
Velocitie Velocitie Pees F1 for help. Details of "Meth" Stat Recording Weid Concetion Pees F1 for help. Details of "Meth" Stat Recording Meth Concetion Pees F1 for help. Diplay Sty Use Geometry Setting Meth Rometring Pees F1 for help. Diplay Sty Use Geometry Setting Meth Rometring Pees F1 for help. Details of "Meth" Node Marge Group Meth Rometring Pees F1 for help. Shere F1 for mat \$1 Node Marge Group Node Marge Group Node Marge Group Expont Profess Saftec Meth Node Marge Group Node Marge Group Node Marge Group Expont Profess Saftec Meth Node Marge Group Node Marge Group Node Marge Group Bandard Spandard Spandard Spandard Spandard Saft Group Goom 0.025 0.500(m) 0.125 0.75		-	Clear Generated Data		Inflation	me of a	sh on sele	ected fac	es.												
Details of "Meth" 12 If the Connection If the Connection Details of "Meth" 3tat Recording Manual Meth Connection If the Connection Display Watch Ret Connection If the Connection If the Connection Display Watch Ret Connection If the Connection If the Connection Display Watch Ret Connection If the Connection If the Connection Display Foremete OFD The Node Marge Group Node Marge Group Bitto Connections Node Marge Group Node Marge Group Node Marge Group Bitto Connections Node Marge Group Node Marge Group Node Marge Group Bitto Connections Node Marge Group Node Marge Group Node Marge Bitto Connections 000 0,250 0,300 (m)		T		*	Weld	I			- 1												
Deske of Medicing Manual Medic Connection K Origing Syle Use Geometry Setting Mach Aldt K Origing Syle Use Geometry Setting Mach Mach Connection K Origing Syle Use Geometry Setting Mach Mach Connection K Origing Syle Use Geometry Setting Mach Mach Connection K Object Peternee FD 10 Mode Marge Group Solver Peternee Flament Older Unear Elsenet Older Unear Node Marge Node Marge Node Marge K Elsenet Older Unear Node Marge Node Marge Node Marge K Bisting Node Marge Node Marge Node Marge K Standard Node Marge Node Marge Node Marge Node Marge Standard Node Marge Node Marge Node Marge Node Marge Standard Node Marge Node Marge Node Marge Node Marge Standard		ар	Kename F2		Mesh Connection	Press F1 f	for help.														
Objeky 0 6 Meh Balt 0 Objeky Sybe Use Geometry Setting 9 Meh Numbering Ordards 9 Meh Numbering 9 Meh Numbering Physics Preference 70 11 Node Marge Group 8 Sher Preference 70 11 Node Marge Group 8 Export Preference 11 12 Node Marge Group 12 Export Preference 10 10 10 10 10 Export Preference Sufface Meth In 0 0000 0,250 0,500(m) 10 9 Sing 0,125 0,75 0,125 <	Details of "Mesh"	_	Start Recording		Manual Mesh Cor	nection K															
Disploy Style Use Geometry Setting Mash Numbering Defaults • Mash Numbering • Mash Numbering Physics Perference CPD ① Node Marge Group Node Marge Group Solver Preference Fluent Node Marge Node Marge Elsenet Oxider Linear Node Marge Node Marge Eport Preview Standard Standard Node Marge Node Marge Bisting Node Marge Node Marge Node Marge Rindsin Uncertains Node Marge Node Marge Node Marge Node Marge Rindsin Uncertains Node Marge Node Marge Node Marge Node Marge Node Marge Node	Display			6	Mesh Edit	^															
○ Paralis 1	Display Style		Use Geometry Setting	-	Mark Numberies																
Physics Perference CFD The Node Marge Group Solver Preference Fluence Fluence <td< td=""><td>Defaults</td><td></td><td></td><td>48</td><td>Mesh Numbering</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></td<>	Defaults			48	Mesh Numbering																
Solver Preference Fluent 6: Node Marge Element Otice Uncer Econof Formie State deta State County County State deta County	Physics Preference		CFD	1	Node Merge Grou	ip 👘															
Element Order Linear Element Order Mode More Eport Provises Laft (57711e-002 m) Node More Eport Provises Visited Heth No Image: Standard Comparison of the Standard Compariso	Solver Preference		Fluent	8:	Node Merge																
Betweet Size Default (70711-020 m) Product move Eponf Formix Standard Standard Standard 0.000 0.250 0.500 (m) Baht Connections 0.125 0.75	Element Order		Linear		No. do Maria																v
Epos of Format Standard Epos of Foreira Standard Standard Epos of Foreira Standard 0 Bolth Concentions 0 Bath Concentions 0	Element Size		Default (7,0711e-002 m)	•	Node Move																1
Export Provises Surface Meth No Stating © Stating © Calaby © Inflation 0.000 0.250 0.125 0.375	Export Format		Standard																		1
Stang Caustry Ca	Export Preview Su	rface I	Aesh No																		
	Sizing																				X
	Quality											0.000		0.250			0.500 (m)				
⊕ Batch Connections 0,125 0,375	Inflation											0,000		0,250			0,500 (m)				
	Batch Connection	5				~							0,125		0,	375	_				

- Right click on $Mesh \rightarrow Insert \rightarrow Face Meshing$:

Figure 9: Face meshing.

- *Geometry* \rightarrow click on *No Selection* and select the cavity \rightarrow click on *Apply*:



Figure 10: Face meshing.

- Check the following settings (Mapped Mesh: Yes, Method: Quadrilaterals):

Scope		
Definition		
Suppressed	No	
Mapped Mesh	Yes	
Method	Quadrilaterals	
Constrain Bounda	ry No	
Advanced		



- Right click on $Mesh \rightarrow Insert \rightarrow Sizing$:

Figure 11: Sizing at the walls.

- Click on the *Edge* icon b \rightarrow select all walls (hold *CTRL* to select multiple edges) \rightarrow click on *Apply*:



Figure 12: Sizing at the walls.

- Set the following settings:
 - Type: Number of Divisions
 - Number of Divisions: 10
 - Behavior: Hard

- Click on the white box next to *Number of Divisions* to set it as a new input parameter **P**:



Figure 13: Sizing at the walls.

- Click on *Generate* to generate the mesh:

File Home Me	text Ish Display Selection	Automation				A : Heate	d cavity - Mesł	ning [CFD	PrepPost -	CFD Base]							Quick La	unch	- & ×
Duplicate Q Insert	Update Generate Mesh	Method	Sizing Face Meshing	Mapped Meshing	Mesh Copy Co	atch ontrol	Contact Sizing Refinement Pinch	Mesh Gi	roup	Connect	Mesh Co	nnection Grou Match Group erge Group	up 💖 Mesh Edit	Mesh Numbering Mesh Edit	Manual Mes Connection	h Contact Match	Bi .do lode Nod erge Mov	He Metric Graph *	Edges etrics Display
Cutline Sam Nmm Sam Image: Sam Sam	demorate the first optimized of the first opt	mesh if out of date. mesh is out of date, sh will be generated.	- + - ×		0		°a Φ + ↔ - - - - - -			Select ×						Cipboard >	[Empty]	Extend -	Select By-
Details of "Mesh" © Dsplay Style © Defaults Physics Preference Solver Preference Element Order © Element Size Encort Format Excort Foreiros Surface Me © Gualty © Gualty © Bath Connections	Vize Geometry Setting CPD Piurnt Default (7,0711e-002 m) Standard h No									0,000	0.125	0,250)	3/2	<u>0,500</u> (m)				Ý • ×

Figure 14: Cartesian mesh for the cavity.

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

2.3 Setup the problem in Fluent

– In WB, right click on $Mesh \rightarrow Update$

- Right click on Setup \rightarrow Edit to run Fluent
- In the *Fluent Laucher* window, tick *Double Precision* \rightarrow *Start*:



Figure 15: Starting Fluent from WB and Fluent settings.

- Tick *Gravity* \rightarrow *Gravitational Acceleration Y* (-*g*) \rightarrow *New Input Parameter* \rightarrow rename the new input parameter as Ra, the Rayleigh number (note the m/s^2 unit)

A:Heated cavity Parallel Fluent@DESKTOP-TG6RB5	9 [2d. dp. pbns. sstkw, 4-processes] [CFD Solver - Level 2 - CFE	O Solver - Level 1 - CFD Basel			- a ×
File Domain Physics User	r-Defined Solution Results View	Parallel Design		Q Qu	ilck Search (Ctrl+F)
Mesh	Zones I	Interfaces Mesh Models	Turbomachinery	Adapt Surfao	e
🛞 Display 👔 👔 Scale	🚱 Combine 👻 📑 Delete 📑 Append 👻 🕎	Mesh Øynamic Mesh	🔲 Turbo Models 🛛 🐇 Turbo Workflow 👻	Manual Statumatic + Create	2 w
🕠 Info 🗸 🧐 😻 🖾 Transform	+ 🖵 Separate + 📅 Deactivate 📸 Replace Mesh	C Mixing Planes	🕞 Turbo Create 🐻 Turbo Topology	🔀 Controls 🔏 Manage 🔗 Manag	30
🕼 Units Check+ Quality + 🕁 Make Polyhedr	ra 🛛 🛷 Adjacency 📑 Activate 🖳 Replace Zone 🗏	Overset 🗺 Gap Model	Periodic Instancing		
Outline View	Task Page	<		Mesh	×
Eliter Test	General	a 📐			A
	Mesh				Ansys
General	Scale Check Report Quality				2022 RI
	Display Units				
Cell Zone Conditions					
Boundary Conditions	Solver Type Velocity Formulation				
KI Mesh Interfaces	Pressure-Based Absolute				
Beference Values	O Density-Based Relative				
Keference Frames					
Named Expressions	Time 2D Space				
Solution Methods	Transient Axisymmetric				
🕺 Controls	Axisymmetric Swirl				
Report Definitions		k.			
Q Monitors	Gravity				
Automatic Mesh Adaption	Gravitational Acceleration				
E Initialization	X [m/s ²] 0				
Calculation Activities Run Calculation	Y [m/s ²] 0				
Results	Z [m/s ²] constant	1			
Surfaces	expression				
Graphics Plots	New Input Param				0 selected all *
Animations	 Named Expressions 	Console			Ø <
Reports Parameters & Customization	New Expression	\FFF.set"			- · · · · · · · · · · · · · · · · · · ·
U rarameters a customization		writing rp variabl	es Done.		

Figure 16: Fluent settings, gravity.

- Change the sign of *Gravitational Acceleration Y* from Ra to -Ra: γ -Ra f_{∞} -
- Models \rightarrow Energy: <u>on</u>, Viscous: <u>Laminar</u>

- *Materials* \rightarrow *Fluid* \rightarrow right click on *air* \rightarrow *Edit*
- Set the following settings:
 - Name: cavity-fluid
 - Density (ρ_0): boussinesq, 1 kg/m³
 - Cp (Specific Heat, c_p): <u>constant</u>, 1 $J/(kg \cdot K)$
 - Thermal Conductivity (k): <u>constant</u>, $1 W/(m \cdot K)$
 - *Thermal Expansion Coefficient* (β): *New Input Parameter* \rightarrow rename the new input parameter as Pr, the Prandtl number (note the K^{-1} unit)
 - *Viscosity* (ν): *Expression*, Pr * 1 [K kg m⁻¹ s⁻¹] (to match units)
- Click on *Change/Create* \rightarrow *Change/Create mixture and Overwrite Air*? \rightarrow *Yes*



Figure 17: Fluid properties.

- Boundary Conditions \rightarrow Wall \rightarrow right click on cold \rightarrow Edit
- Thermal \rightarrow Temperature (T_c): 0 K (negative temperatures are not allowed)



Figure 18: Isothermal boundary condition for the *cold* wall.

- Repeat the same for the *hot* wall: *Temperature* (T_h) : 1 K

- Physics \rightarrow Operating Conditions \rightarrow Operating Temperature ($T_0 = (T_h + T_c)/2$): 0.5 K



Figure 19: Setting the operating temperature.

- Setup \rightarrow Reference Values \rightarrow set Length to 1 m and Temperature to 0 K.

```
-Console \rightarrow type solve set expert
```

```
– Allow selection of all applicable discretization schemes? \rightarrow type yes
```

- Methods \rightarrow Spatial Discretization \rightarrow select Central Differencing for both Momentum and Energy:



Figure 20: Setting the discretization schemes.

- *Controls* \rightarrow set all relaxation factors to 1
- *Limits*... \rightarrow set all minimum values to 0

A:Heated cavity Parallel Fluent@DESKTOP-TG6RB5	i9 [2d, dp, pbns, lam, 4-processes] [CFD Solver - Level 2 - CFD Solver - Level 1 - CFD Base]	- 🗆 ×
🖉 🎕 🎕 H I I 🎆 🎞 🖺		
<u>File Domain Physics User-</u>	- Defined Solution Results View Parallel Design 🔺 🔍 Quick Search (Cirlis	🕫 🚺 🚺
Solver Solver Solver Solver Solutions	Models Materials Zones Model Specific Is Rotation Multiplase 41 Structure Cell Cell Specific Is weak Exhanger Xi Species KM Accords v Endations Creative Exhanger Xi Species Model Specific Endations Creative Exhanger Xi Species Model Specific Endations Creative Exhanger Xi Species Model Specific Endations	
Outline View <	Task Page Mesh	×
Filter Text	Solution Controls	
B General Control Control Contro Contro Control Control Control Control	Image: Control in the second secon	
Controls Report Definitions Report Definitions Report Definitions Report Reports Report Reports Report Rep	Default Immunikative Pressure [Pa] 0 Equations Immunikative Pressure [Pa] 0 Minimunikative Pressure [Pa] 0 Immuni	0 selected at • •

Figure 21: Setting the relaxation factors and lower limits for pressure and temeprature.

- Right click on *Report Definitions* \rightarrow *New* \rightarrow *Surface Report* \rightarrow *Area-Weighted Average...* for the computation of the mean Nusselt number \overline{Nu} :

- $Name \rightarrow nu$
- Field Variable \rightarrow Wall Fluxes... \rightarrow Surface Nusselt Number
- Surfaces \rightarrow hot
- tick Create Output Parameter

A:Heated cavity Parallel Fluent@DESKTOP-TG6R	1859 [2d, dp, pbns, lam, 4-processes] [CFD Solver - Level 2 - CFD Solver - Level 1 - CFD Base]	-		:
🛛 🗶 🌒 z 🦸 🎆 🖽 🞬				
Elle Domain Physics Us	eer-Defined Solution Results View Parallel Design + 🔍	F) 📀		ys
Solver Operating Conditions Ceneral Planar Energy	Models Materials Zones Model Specific Radstone ■ Multiples ↓ Structure □ Colored Hase □ Colored Hase > Diplet Thermal Coupling ↓: Network Dictionary Colored Hase O More □ Colored Hase > O TOM Rays ↓: > Discrete Hase O More □ Prodels □ Directer Hase > O TOM Rays ↓: > Discrete Hase O More □ Prodels □ Directer Hase > O TOM Rays		×	
Outline View	< Task Page < Task Page Benot Type		;	×
Filter Text	Reference Values Options Optio		ısy	S
Satury Satury Satury Satury Satury Control Report Defendions Carrol Control Control	Reference Values Custom Vectors Areage Over 1 Denny (typ/m) 1 0 Suffice Koseth Humber Suffice Koseth Humber Suffice Koseth Humber Suffice Koseth Humber			RI
Cell Registers New Automatic Wesh Ada Expand All Collapse All Collapse All Collapse Raults Rasults	Logression			
 Yarameters & ubsome dom Simulation Reports 	Liser Defined. Fact Average.		_	Ţ
	Standard Devaldon Sum Unformity index - Area Wrighted Unformity index - Area Wrighted Unformity index - Area Wrighted Verter Average Verter Average Verte		<u>A</u> 0	×

Figure 22: Computation of \overline{Nu} .

- Residual \rightarrow Show Advanced Options \rightarrow Convergence Criterion \rightarrow none since we'll use a fixed number of iterations:

A:Heated cavity Parallel Fluent@DESKTOP-TG6RB5	59 [2d, dp, pbns, lam, 4-processes] [CFD Solver - Level 2 -	CFD Solver - Level 1 - CFD Base]				- 🗆 ×
 응 환 값 값 분 후 🧖 표 발 File Domain Physics User 	-Defined Solution Results View	r Parallel Design	*		Q Quick Search (Ctri+E)	
Mesh Display Dinfo + Wesh Display Dinfo + Wits Pulits Check+ Quality + Make Polyhedr	Zones	Interfaces Mesh Models Image: Mesh Image: Dynamic Mesh Image: Mesh Image: Dynamic Mesh Image: Overset Image: Gap Model	Turbomachinery Turbo Models Strubo Workflow Turbo Create C Turbo Topology Speriodic Instancing	Adapt Manual Automatic Controls Manage	Surface + Create ↓ Manage	
Outline View <	Task Page General			Mesh		× Ansvs
Setup Subtain Su	Residual Monitors Options Freit Consels Residual Monitors Prict Consels Residual Monitor Residual Residual	Perioda Monito Perioda Monito Perioda Monito Perioda Monito Perioda Security Perioda Secu	5 Equations Resolut revelory revelory revelory Convergence Conditions ✓ Show Abaned Options Resolut Value Scale © compute Lood Scale Resourceable	Contergence Orterion		2022 m
		Console Setting parallel, Done.	OK Plot Cancel Help		0 selecte	d all v

Figure 23: Convergence conditions.

- Initialization \rightarrow Initialize
- Double click on *Calculation Activities* \rightarrow *Create/Edit...* to save the results to file:
 - Defined Commands: 1
 - tick on <u>Active</u>
 - Every: <u>250 Iterations</u>
 - Command: file/export/ascii cell_center_values.txt () yes temperature pressure y-velocity x-velocity () yes

A:Heated cavity Parallel Fluent@DESKTOP-TG6RB5	59 [2d, dp, pbns, lam, 4-processes] [CFD Solver - Level 2	CFD Solver - Le	vel 1 - CFD Base]					- 🗆 ×
🖌 🎕 🛍 a 🦸 🎆 🖬 🖀								
File Domain Physics User-	-Defined Solution Results Vie	w Pa	urallel Desig	^			Q Ouick Search (C	arl+F) 💿 🥫 Ansys
Mesh Display	Zones	Interfaces	Mesh Models Dynamic Mesh Mixing Planes Gap Model	Turbo Turbo Models	machinery ∯ Turbo Workflow ↓ ᠿ Turbo Topology ∯ Periodic Instancing	Adapt Manual Sin Automatic Controls & Manage	Surface + Create ↓ & Manage	
Outline View <	Task Page	<	*			Mesh		×
Filter Text Solution Solution Methods Controls Report Definitions Controls Cont	Calculation ActiVities Autosve Every (Berstons) 0 \$ Catt Automatic Deport							Ansys 2022 RI
Adomate Mein Aughon Caluadori Adomes Caluadori Adomes Parameters & Customization Simulation Reports	Create y Edit. Delete Execute Commands	Execute C Defined Comma Active Name Save-to comma	commands nds 1 Every 250 nd-2 1 nd-3 1	When theration × theration ×	Command file/export/ascii cell_cerr	ter_values.bt () yes temperature press	sure y-velocity x-vek	
	Cresta/Edt) Automatically Initialize and Modify Case		nd-5 1	(Iteration * (Iteration * (Iteration * (Iteration * (Iteration * (K) Del	ine Macro Cancel	нер		

Figure 24: Save results to file.

- Run Calculation \rightarrow Length Scale Method: <u>Aggressive</u>, Number of Iterations: <u>250</u>

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

- In WB, double click on *Parameter Set* and specify the values of the parameters:

- Edge Sizing Number of divisions: 10,20,40,80,...,640
- Ra: <u>10⁵</u> (1e5)
- Pr: <u>0.71</u> (0, 71)

😸 HC - Workbench														-		×
File Edit View Tools Units Exte	nsions	Jobs Help														
11 12 12 C																
🚹 🚰 😹 🕕 Project 🖗 Par	ameter	Set ×														
Resume 77 Update All Design Points																
Toolbox 👻 🕂 🗙	Outine	of All Parameters			₹ д	x Tal	ble of t	Design Points								- q x
		A	в	с	D			A	в	с	D	E	F	G	н	
No toolbox items are applicable for the	1	ID	Parameter Name	Value	Unit				P1-Edge	_			_	Patainad	_	
current selection.	2	Input Parameters					1	Name	Sizing Number	P2-Ra 💌	P3 - Pr	💌 P4-Nu 💌	Ret	Data	Note 💌	4
	3	Heated cavity (A1)					2	Units		m s^-2 ▼	K^-1	-				11
		15 D1	Edge Sizing	10			3	DP 0 (Current)	10	1E+05	0,71	5,3842	V .	~		1
	1	ψ *1	Divisions	10			4	DP 1	20	1E+05	0,71	4,8907	V	~		-
	5	Ф Р2	Ra	1E+05	m s^-2 💌		5	DP 2	40	1E+05	0,71	4,619	V	 		
	6	ф РЗ	Pr	0,71	K^-1 🔄		6	DP 3	80	1E+05	0,71	4,5466	V	 		
	*	New input parameter	New name	New expression			7	DP 4	160	1E+05	0,71	4,528	v	 		
	8	 Output Parameters 					8	DP 5	320	1E+05	0,71	4,5232	v	 		
	9	🖃 🙆 Heated cavity (A1)					9	DP 6	640	1E+05	0,71	4,522		 		
	10	P4 ⊊	Nu	5,3842			•									
	-	New output parameter		New expression												-
	12	Charts				1										
View All / Customize	<					>				Iob	Monitor	No DPS Connection	Show Pr	ogress 🚅	Show 2 Mes	sages

Figure 25: Parameters.

- Click on Update all Design Points to run the simulations for the whole set of parameters

- The values of the mean Nusselt number \overline{Nu} are displayed as output parameters

- Using file manager (File explorer/Finder/etc.) go to the project directory, i.e., where your WB project file HC.wbpj has been saved

- HC_files\dp0\FFF\Fluent is the location of the files for Design Point 0 (DP0)

- cell_center_values.txt is the formatted text file containing the results of the simulation, i.e., the variables at the centroid of each cell:

cellnumber,	x-coordinate,	y-coordinate,	x-velocity,	y-velocity,	pressure,	temperature	
1,	9.50000000E-01,	5.00000000E-02,	-1.046026133E+01,	-5.431995278E+01,	3.941059612E+03,	3.813460988E-02	
2,	9.50000000E-01,	1.50000000E-01,	-1.333146451E+01,	-2.096147773E+01,	5.525523339E+02,	9.241430683E-02	
3,	8.50000000E-01,	5.00000000E-02,	-2.880271365E+01,	-4.176336734E+01,	3.287822903E+03,	9.738570710E-02	
4,	9.50000000E-01,	2.50000000E-01,	-5.669191625E+00,	-6.513594303E+01,	-1.885156534E+03,	1.616692471E-01	
5,	8.50000000E-01,	1.50000000E-01,	-3.219779372E+01,	-4.399573009E+00,	-5.219206826E+02,	2.040169185E-01	
6,	7.50000000E-01,	5.00000000E-02,	-4.225412273E+01,	-3.020070824E+01,	2.391560976E+03,	1.303767968E-01	
7,	9.50000000E-01,	3.50000000E-01,	-5.073024206E+00,	-6.853919810E+01,	-3.819253019E+03,	2.138247619E-01	
8,	8.50000000E-01,	2.50000000E-01,	-1.917700808E+01,	-4.329798046E+01,	-2.604515360E+03,	3.207748617E-01	
9,	7.50000000E-01,	1.50000000E-01,	-4.421236949E+01,	1.471949386E+01,	-1.109332743E+03,	2.322885827E-01	
10,	6.50000000E-01,	5.00000000E-02,	-3.695429154E+01,	-2.769231322E+01,	1.852409333E+03,	1.491388631E-01	
11,	9.50000000E-01,	4.50000000E-01,	5.003548954E-01,	-7.630208568E+01,	-4.496550953E+03,	2.659435075E-01	

3 MATLAB post-processing

3.1 Structure of the main script

In MATLAB we'll read the results of the simulations in order to compute:

- the Nusselt number along the hot wall, Nu_y , equation (15)
- the mean Nusselt number, \overline{Nu} , equation (16)
- the streamfunction, ψ , as the solution of $\nabla^2 \psi = -\omega$
- the maximum value of $|\psi|$, i.e., ψ_M , and the corresponding location (x_M, y_M)

The structure of the main MATLAB script (that should be put in the same directory of the WB project file HC.wbpj) is the following:

```
PostProcessing.m
```

```
addpath('MATLAB_functions') ;
1
2
   design_points = 0:6 ;
3
   results_variables = {'Nu', 'psi_M', 'x_M', 'y_M'};
4
   results = initialize_table(design_points, results_variables) ;
5
6
   for dp = design_points
7
8
9
      % Reading the ascii formatted file of 2D results
     CCV_file = sprintf('HC_files\\dp%d\\FFF\\Fluent\\cell_center_values.txt', dp) ;
10
             = readmatrix(CCV_file) ;
     CCV
11
12
      [~, x, y, u, v, p, T] = <u>extract_columns</u>(CCV) ;
     K = matrix_of_linear_indices(x,y) ;
13
14
     [ x, y, u, v, p, T] = <u>vector_to_matrix</u>(K, x, y, u, v, p, T);
15
      % Nuy = -dT/dx at the hot (left) wall
16
     Nuy = -(T(1,:) - 1) / (x(1,1) - 0);
17
18
      % Mean Nu
19
     results.Nu(dp+1) = mean(Nuy) ;
20
21
2.2
     % Streamfunction
23
     w = vorticity(x, y, u, v);
     psi = Poisson_2D_solver(x, y, -w) ;
24
25
     % Maximum of |psi|
26
27
     [psi M, k]
                          = \max(abs(psi(:)));
     results.psi_M(dp+1) = psi_M ;
28
     results.x_M(dp+1) = x(k);
29
     results.y_M(dp+1) = y(k);
30
31
   end
```

- Line 1: definition of the directory where the required <u>user-defined functions</u> are stored, i.e., MATLAB_functions\.

- Lines 3-5: we define the design points to read, the names of the quantities that we want to compute for each design, and then we initialize the table used to store these quantities.

- Lines 7-31: the main for loop over each design point.

- Lines 10-14: we read the formatted text file containing the results, i.e., the coordinates x and y of the centroids of the cells, the velocities u and v, pressure p and temperature T at the centroids. Some manipulation is required to rearrange the unstructured order of the cells in Fluent into structured data, i.e., 2D matrices. After line 14, x, y, u, v, p and T are $N_x \times N_y$ matrices, where N_x and N_y are the number of cells along x and y, respectively.

- Line 17: computation of the vector Nu_y of the values Nu_y at the hot wall through a finite difference, equation (15).

17

- Line 20: computation of \overline{Nu} through a simple average of vector Nuy, equation (16), and storage in the results table.

- Lines 23-24: computation of vorticity ω and streamfunction ψ at cell centers.

- Lines 27-30: computation of the maximum value of $|\psi|$ and its corresponding location, and their storage in the results.

3.2 Some user-defined functions

The most important user-defined functions employed in the main script are listed as follows.

matrix_of_linear_indices: this function takes the vectors x and y of the coordinates of the cell centroids and returns the matrix K such that, given a vector v of any flow variable computed by Fluent, v (K) is the corresponding structured matrix. K is therefore the $N_x \times N_y$ matrix containing the indices of the cells in structured order. The vectors x and y must be obtained from a cartesian structured grid (not necessarily uniform nor square):

matrix_of_linear_indices.m

```
1
   function K = matrix_of_linear_indices(x,y)
     [y, K] = sort(y) ;
2
           = find(y-y(1) > 1e-6, 1) - 1;
3
     Nx
     Ncells = length(y) ;
4
          = Ncells / Nx ;
5
     Ny
           = reshape(K, Nx, Ny) ;
6
     for j = 1 : Ny
7
       x_j = x(K(:,j));
8
       [\sim, i] = sort(x_j);
9
       K(:, j) = K(i, j);
10
11
     end
   end
12
```

- Line 2: the vector y is sorted with increasing order, i.e., the cells are sorted along y. K is the vector of indices for this sort. Since the grid is structured, we are now left with N_y horizontal rows of cells, each of which requires a sort along x.

- Line 3-5: the number of cells N_x and N_y along x and y is computed together with the total number of cells N_cells = $N_x \cdot N_y$.

- Line 6: K is reshaped into a $N_x \times N_y$ matrix in order to x-sort each row of cells, corresponding to matrix columns, separately from one another.

- Line 7-11: the for loop over each row of cells.

- Line 8-10: each row of cells is sorted with increasing order along x, modifying matrix K accordingly.

vorticity: this function takes the matrices of coordinates x,y and the matrices of velocities u,v and computes the matrix w of the vorticity ω at the cell centers by using central finite differences with constant step $2\Delta x$ along x and $2\Delta y$ along y:

```
vorticity.m
```

```
function w = vorticity(x, y, u, v)
    dx = x(2,1) - x(1,1); % only for uniform grids, ie, dx = constant
2
    dy = y(1,2) - y(1,1); % only for uniform grids, ie, dy = constant
4
     [u, i, j] = add_ghost_cells(u);
5
     [v, i, j] = add_ghost_cells(v) ;
6
7
```

1

3

```
% w = dv/dx - du/dv
8
     w = (v(i+1,j) - v(i-1,j)) / (2*dx) - (u(i,j+1) - u(i,j-1)) / (2*dy);
9
10
   end
11
   function [Z, i, j] = add_ghost_cells(M)
12
13
     [Nx, Ny] = size(M);
     Z = zeros(Nx+2, Ny+2); % the new matrix with ghost cells
14
15
     i = 2 : (Nx+1) ;
     j = 2 : (Ny+1) ;
16
     Z(i,j) = M ;
                              % original cells
17
     Z(i,1) = -Z(i,2); % ghost cells, south side, 0 Dirichlet
18
     Z(i,end) = -Z(i,end-1); % ghost cells, north side, 0 Dirichlet
19
     Z(1, j) = -Z(2, j); % ghost cells, west side, 0 Dirichlet
2.0
21
     Z(end,j) = -Z(end-1,j) ; % ghost cells, east side, 0 Dirichlet
   end
22
```

- Line 2-3: the (constant) cell size $[\Delta x, \Delta y]$ is obtained from the coordinates x and y.

- Line 5-6: ghost cells are added to matrix of velocities u and v according to zero Dirichlet boundary condition u = 0 at the walls. This operations is done by <u>add_ghost_cells</u> defined at lines 12-22. Ghost cells are added in order to simplify the computation of ω at the boundary cells.

- Line 9: the matrix w of vorticity is computed through central finite differences in a single line by using vector indexing through vectors i and j, allowing therefore block matrix operations.

- Lines 12-22: function [Z, i, j] = <u>add_ghost_cells</u> (M) adds ghost cells to matrix M, returning Z. This function returns also vectors i and j containing the indices for the internal cells in matrix Z.

- Lines 13-14: the new matrix with 2 additional rows and columns is initialized.

- Lines 15-16: vectors i and j contain the row and columns indices of the internal cells in matrix Z: Z(i, j) is therefore the original matrix M, which is copied in the same positions of Z at line 17.

- Lines 18-21: the values of ghost cells are set according to zero Dirichlet boundary conditions. This is done by using vectors i and j for each side of the square boundary.

<u>Poisson_2D_solver</u> (x, y, b): a FV solver for a Poisson equation on a cartesian $N_x \times N_y$ uniform rectangular grid, i.e., Δx and Δy are constant but not necessarily equal. x, y and b are the $N_x \times N_y$ matrices of x and y coordinates of the centroids of the cells, and right hand side term b at the centroids, respectively. This solver has already been illustrated during a previous MATLAB lesson on solving PDEs.

3.3 Results

By running the main script PostProcessing.m, the results can be displayed by typing results in the MATLAB Command Window:

>> results results = 7x4 <u>table</u>			
Nu	psi_M	x_M	у_М
5.3842	8.1345	0.35	0.65
4.8907	9.1764	0.725	0.375
4.619	9.5061	0.7125	0.3875
4.5466	9.5884	0.71875	0.39375
4.528	9.6101	0.28437	0.60313
4.5232	9.6151	0.28594	0.60156
4.522	9.6164	0.28516	0.60078

These results can be compared with the benchmark values obtained in [2]: $\overline{Nu} = 4.519$, $\psi_M = 9.612$ at $(x_M, y_M) = (0.285, 0.601)$. We can also see that the \overline{Nu} values, Nu, are identical to those directly computed by Fluent, Figure 25.

Since the simulations have been performed on a sequence of grids obtained by doubling the number of cells along each dimension, i.e., with grid sizes $h, h/2, h/4, h/8, \ldots$, the Richardson extrapolation can be easily applied to any computed quantity A:

$$\tilde{A}(h) = A + Ch^p + O(h^{p+1})$$
(17)

where $\hat{A}(h)$ is the approximation of A computed on a grid with size h, p is the convergence rate of the employed method (p = 2 for the standard second order FV discretization, as the one employed by Fluent) and C is an unknown constant. By writing equation (18) for a grid size h/2 and solving for A, we obtain the Richardson extrapolation:

$$A = \frac{2^{p}\tilde{A}(h/2) - \tilde{A}(h)}{2^{p} - 1} + O(h^{p+1})$$
(18)

which has convergence rate p + 1. The process can then eventually be iterated to obtain convergence rates p + 2, p + 3, ... The MATLAB implementation of the iterative Richardson extrapolation in this case is quite easy, here applied to $A = \overline{Nu}$:

RichardsonExtrapolation.m

```
Richardson = @(v,p) ( 2^p * v(2:end) - v(1:end-1) ) / ( 2^p - 1 ) ;
A = results.Nu;
for p = 2:4
    A = Richardson(A,p)
end
```

which gives the following extrapolations:

p=2	p = 3	p = 4
A = 4.726242 4.528405 4.522511 4.521764 4.521653 4.521638	A = 4.500142 4.521670 4.521657 4.521637 4.521636	A = 4.523105 4.521657 4.521636 4.521636

20

It is possible to verify the convergence rate p = 2 of the results obtained with Fluent by computing the error between the \overline{Nu} values and their best approximation available, i.e., the last value obtained by the Richardson extrapolation:

A_best = A(end) ;
FV_error = abs(results.Nu - A_best) ;

Since we want to plot the convergence curve, the grid size $h = 1/N_x$ is needed:

Nx = 10 \star 2 .^ design_points ; h = 1 ./ Nx ;

A reference 2^{nd} order curve is useful for graphical comparison:

```
II_order = FV_error(end) * (h/h(end)).^2;
```

Plot of both curves with log-log scales, with some graphical embellishment:

```
hold on :
plot(h, FV_error, 'o-b', 'LineWidth', 1);
plot(h, II_order, '--k', 'LineWidth', 1);
ax = gca ;
ax.XScale = 'log' ; ax.YScale = 'log' ;
ax.XGrid = 'on'; ax.YGrid = 'on'
                                         ;
ax.XLabel.String = '$h$' ;
ax.YLabel.String = 'Error(Nu)' ;
ax.TickLabelInterpreter = 'latex'
                                    ;
ax.XLabel.Interpreter = 'latex' ;
ax.YLabel.Interpreter
                         = 'latex' ;
ax.Box = 'on' ;
ax.FontSize = 18 ;
leg = legend('Fluent', '$2^{nd}$ order', 'Location', 'northwest');
leg.Interpreter = 'latex' ;
exportgraphics(gcf, 'Error_Nu.pdf') ;
```

which produces the following figure, confirming the 2^{nd} order accuracy of Fluent:



Figure 26: Convergence curve for \overline{Nu} .

References

- [1] G. De Vahl Davis. Natural convection of air in a square cavity: A bench mark numerical solution. *International Journal for Numerical Methods in Fluids*, 3(3):249–264, 1983.
- [2] G. De Vahl Davis and I. P. Jones. Natural convection in a square cavity: A comparison exercise. *International Journal for Numerical Methods in Fluids*, 3(3):227–248, 1983.
- [3] P. Le Quéré and M. Behnia. From onset of unsteadiness to chaos in a differentially heated square cavity. *Journal of Fluid Mechanics*, 359:81–107, 1998.