Enrico Nobile, Riccardo Zamolo

DIA - Dipartimento di Ingegneria e Architettura University of Trieste

Esercitazioni di Termofluidodinamica Computazionale

Simulation of the flow in a 90° elbow with ANSYS Fluent





April 2025

1 Introduction

In this document, all the steps necessary to solve a fluid dynamic problem, whose geometry is particularly simple, will be described in detail using *ANSYS Fluent 2024R1*[®] (hereinafter Fluent for brevity). To this end, we will use *ANSYS Workbench*[®] (WB hereinafter), whose characteristics make it particularly convenient and easy to understand the sequence of steps used for any CFD analysis. In other tutorials, WB will not be used, for convenience, although its use is always possible.

The chosen problem, as mentioned, is particularly simple from a geometric point of view, and this will allow us to define the geometry directly within WB via *ANSYS Design Modeler*[®] (DM hereafter), rather than importing the geometry generated through an external CAD. However, it is a problem with interesting characteristics:

- Flow with non-trivial peculiarities (secondary motions);
- Wide availability of experimental data, global and local;
- Accuracy, qualitative and quantitative, of the predictions strongly influenced by the analysis methods.

It is perhaps unnecessary to point out that there are numerous analogous cases in the literature, characterized by simple geometry and non-trivial or complex motion characteristics, which therefore lend themselves well as a topic for the *student project*.

2 Description of the problem

We wish to study the turbulent flow, for an incompressible fluid (water at 20 °C) and in the absence of heat transfer, through a 90° bend (elbow) of a circular section pipe, as shown in figure 2.

The problem data are summarized in the table 1.

Characteristic data of the problem.						
D	=	50 mm				
L_1	=	200 mm				
L_2	=	300 mm				
r_i	=	100 mm				
r_e	=	150 mm				
U_m	=	0.2 m/s				
μ	=	$8.899 \times 10^{-4} \text{ kg/(m s)}$				
ρ	=	997 kg/m ³				
Re	=	$\frac{\rho U_m D}{\mu} = 9980$				

Table 1: Data for the 90° elbow.



Figure 1: Characteristic dimensions of the 90° elbow.

In past experimental studies (H. Nippert, 1929¹) for this problem, it was observed:

- 1. Possible presence of separation phenomena, both on the extrados and on the intrados, in the latter case downstream from the point of greatest curvature;
- 2. Superposition, on the main motion, of a secondary current, which tends to shift the maximum values of the velocity towards the external side, and whose origin can be explained as follows. If the motion were irrotational, the velocity profile, uniform in the straight section, in the curved section would present greater values towards the inside (convexity) and lower towards the outside (concavity). According to the principle of conservation of mechanical energy (Bernoulli's theorem), therefore, there would be a progressive transformation of kinetic energy into pressure energy along the external wall, and an opposite transformation for the internal wall. As a result of friction, however, the fluid near the walls is slower than that of the irrotational model, and therefore both the pressure recovery on the external side, and the pressure drop on the internal side, will be lower than the theoretical values. Compared to the situation predicted by the irrotational model, there will be an imbalance of pressures which determines a certain transfer of fluid from the inside to the outside of the curve.
- 3. The secondary current that is thus determined tends to shift the maximum values of the velocity towards the external side, as illustrated in the experimental findings in figure 2, and to create a double spiral motion that has important consequences: in addition to producing a further dissipation of energy, these secondary motions extend for a very long stretch after the end of the curve even 40 60 diameters and can significantly distort the measurements, carried out with venturi meters, counters etc., placed at an insufficient distance from the curve itself.

¹Nippert, H. 1929, Uber den Strömungswiderstand in gekümmten Kanalälen. Forschungsheft Arbeit Ingenieur-Wesen, no. 320.



Figure 2: Characteristics of the flow in a 90° elbow detected experimentally.

3 Analysis with ANSYS Workbench and Fluent

• Start ANSYS Workbench (WB for brevity) via the \Re icon on the desktop, or via $Start \rightarrow ANSYS \ 2024 \ R1 \rightarrow Workbench \ 2024 \ R1$

The WB interface will look like the one in figure 3.

🗱 Unsaved Project - Workbench		- 0	\times
File View Tools Linits Extensions John Help			
🚹 💕 📓 💐 🗍 Project			
👔 Import 🍣 Reconnect 🔞 Refresh Project 🖉 Update Project 🔛 ACT Start Page			
Todax • 0 x Project Schematic			- # X
En countad tial diamonic			
Coupled Field Medd			
Coupled Field Water			
Conjunct Field Transient			
Coopervalue Burklinn			
Explicit Dynamics			
Fluid Flow (CFX)			
G Fluid Flow (Fluent with Fluent Meshing)			
Fluid Flow (Fluent)			
G Fluid Flow (Polyflow)			
Harmonic Acoustics			
Marmonic Response			
2 Hydro dynamic Diffradion			
Reg Hydrodynamic Response			
🗵 LS-DYNA			
N LS-DYNA Restart			
i Magnetostatic			
Te Modal			
[20] Modal Acoustics			
with a second se			
3 Random Vibration			
100 Response Spectrum			
Rigid Dynamics			
Speos			
Static Acoustics			
Zabic Structural			
Cleary-state memai			
Transient Structura			
Turbuscher Fluid Flow			
E Component Systems			
Y Vew All / Customize			
Ready	🔜 Job Monitor 🕎 No DPS Connection 🔛 No HPC Platform Services Connection 📼 Show Progress 🔅	Show 0 Mess	sages

Figure 3: Starting ANSYS WB.

• Select *SI* or *metric units* of measurement from the *Units* menu as shown in figure 4. Note that it will always be possible to change this choice later, even customizing the



values of the units of measurement for each quantity, from the WB Units menu.

Figure 4: Selecting measurement units in ANSYS WB.

• Select, from the *Toolbox* list on the left, in the *Analysis Systems* group, *Fluid Flow* (*Fluent*) and drag it into the *Project Schematic* pane, as shown in figure 5.

2 Unsaved Project - Workbench	>	<
File View Tools Units Extensions Jobs Help		
🗋 💕 🖩 💐 🕕 Project		
👔 Import 🗟 Reconnect 👔 Refresh Project 🥖 Update Project 📲 ACT Start Page		
Toobox v 0 x Project Schematic	- D	x
Coupled Field Other		
Copreducting Hamabala 2 C Geometry 2		
General Control Contro		
W Explicit Dynamics 4 🍓 Setup 👕 🖌		
G Fluid Flow (CFX) 5 Solution		
Regults		
G Fluid Flow (Fluent)		
Fluid Flow (Polyriow)		
E3 Harmonic Acoustics		
Marmonic Response		
2 Hydrodynamic Diffraction		
Research Hydrodynamic Response		
E LS-DYNA		
😢 LS-DYNA Restart		
00 Magnetostatic		
👜 Modal		
B Modal Acoustics		
Reg Motion		
2 Random Vibration		
Response Spectrum		
Rigid Dynamics		
Speos		
Date Acoustics		
Court Scructuron		
Security and the second s		
Ne Substructure Generation		
Thermal-Electric		
Transient Structura		
Transient Thermal		
Turbomachinery Fluid Flow		
Component Systems		
ACP (Post)		
ACP (Pre)		
I view Ar / Customize		_
Ready	🔝 Job Monitor 🖳 No DPS Connection 📃 No HPC Platform Services Connection 📼 Show Progress 😕 Show 0 Messages	1.3

Figure 5: Selection of Fluent in ANSYS WB.

• The system in the Project Schematic pane will now appear as in figure 6, in which you can notice that the various items, called cells (*cells*) in WB, which make up the project scheme - *Geometry, Mesh, Setup, Solution* and *Results* - are distinguished by a question mark, to signify the absence of data or the presence of errors.

•	А						
1	S Fluid Flow (Fluent)						
2	Geometry	? 🖌					
3	🍘 Mesh	? 🖌					
4	🍓 Setup	2 🖌					
5	Solution	2 🖌					
6	🥩 Results	? 🖌					
	Fluid Flow (Fluent)						

Figure 6: Cells status in the Fluent project.

It will now be necessary to proceed, from top to bottom, starting from *Geometry* and, once completed that step, move on to *Mesh*, and so on. Every time a phase (cell) is completed correctly, it will appear, instead of a question mark, with a green *checkmark* symbol (\checkmark).

At this point it is appropriate to describe the different *types* and *states* of the *cells*, referring to the *Help* for further information.

Types of cells

The most common *types of cells* that occur in many of the analysis and component systems available in Workbench are:

Engineering Data

Use the *Engineering Data* cell with *Mechanical systems* or the *Engineering Data* component system to define or access material models for use in an analysis.

Geometry

Use the *Geometry* cell to import, create, edit or update the geometry model used for analysis. Right-click the cell to access these functions in the context menu.

Mesh

The *Mesh* cell in Fluid Flow analysis systems or the Mesh component system is used to create a mesh using the Meshing application. It can also be used to import an existing mesh file.

Setup

Use the *Setup* cell to launch the appropriate application for that system. You select solution methods and algorithms and define materials and boundary conditions. The data from the application is incorporated into the project in Workbench, including connections between systems.

Solution

From the *Solution*² cell, you can access the Solution branch of your application, and you can share solution data with other downstream systems.

²Selecting Edit from the *Setup* cell loads grid and settings and *not* the current case/data file or the initial data file. To load the current or initial data file, select Edit from the *Solution* cell instead.

Results

The *Results* cell indicates the availability and status of the analysis results (commonly referred to as *postprocessing*). From the Results cell, you cannot share data with any other system.

Typical Cell States

P Unfulfilled

The required upstream data does not exist. Some applications may not allow you to open them with the cell in this state. This is the case, for example, if you have not yet assigned a geometry to a system: all downstream cells appear as unfulfilled, because they cannot progress until you assign a geometry.

Refresh Required

The upstream data has changed since the last *Refresh* or *Update* and it may or may not be necessary to regenerate the output data. When a cell is in this state there are generally several options:

- Edit and review the unrefreshed data.
- *Refresh* the data, but without incurring any time-consuming operations, such as generating a new mesh if the geometry has changed.
- Update the cell, which will result in a *Refresh* and *regeneration of the data*, such as generating a new mesh if the geometry has changed.

The advantage of being able to perform a *Refresh*, instead of an *Update*, concerns the possibility of obtaining diagnostic information on the downstream cells without performing a real *Update*, potentially much more expensive in terms of time and computational resources.

? Attention Required

All of the cell's inputs are current. However, a corrective action has to be taken in order to proceed. To complete the corrective action, you may need to interact with this cell or with an upstream cell that provides data to this cell. Cells in this state cannot be updated until the corrective action is taken. This state can also signify that no upstream data is available, but you can still interact with the cell. For instance, some applications support an *empty* mode of operation in which it is possible to enter the application and perform operations regardless of the consumption of upstream data. This is the case, for example, of the *Geometry* cell.

Update Required

The local data of the cell has been modified and therefore the output of the cell must be regenerated. When updating a cell that requires *Refresh*, the *Refresh* operation will be performed first, followed by *Update*.

🗹 Up to Date

An update has been performed on the cell and no failures have occurred. It is possible to edit the cell, and for the cell to provide up-to-date data to other cells.

Input Changes Pending

The cell is up to date, but its state may change as upstream cells are updated.

Solution-Specific States

The Solution or Analysis cell for certain solvers can display the following solutionspecific states.

* Interrupted, Update Required

The solution has been stopped *correctly*: the solver has completed its iteration and produced a *solution file*. This file can be used for post-processing - for example to verify that the solution is proceeding towards a physically correct result - or to restart the solution, using the *Resume* or *Update* functions..

፳ Pending

Signifies that a batch or asynchronous solution is in progress. When a cell enters the pending state, you can interact with the project to exit Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell is not in an up-to-date state when the solution completes.

Failure States

If a particular action fails, Workbench provides a visual indication as well. You can pane any related error messages in the *Messages* pane by clicking *Show Messages* in the lower right portion of the window.

Refresh Failed, Refresh Required

The last attempt to refresh cell input data failed and the cell remains in a refresh required state.

🗴 Update Failed, Update Required

The last attempt to update the cell and calculate output data failed and the cell remains in an update required state.

3 Update Failed, Attention Required

The last attempt to update the cell and calculate output data failed. The cell remains in an attention required state.

In the meantime, let's save the WB project in the chosen working folder (it is advisable to use a separate working folder for each project), by selecting the *Save Project as* icon
 If from the menu bar or from *File* → *Save as*, and name it *ElbowFluent* (the extension .wbpj, from WorkBench ProJect, will be assigned automatically), as in figure 7.

3.1 Geometry generation

- In geometry generation, there are more ways to reach the same result: the choice, in addition to being dictated by experience, depends on numerous factors, such as:
 - Ease of subsequent grid generation, and quality of the latter;
 - Geometry parameterization, for subsequent parametric analysis, optimizations, etc.;
 - Export of the results to other products and/or other physics (e.g. fluid-structure interactions).

Unsaved Project - Workbenck	h								
File View Tools Units E	extensions Jobs Help	Save As							×
New Open	Ctrl+N Ctrl+O	$\leftarrow \rightarrow ~ \checkmark ~ \uparrow$	📒 « Termofluid	_Comput > Esercita	azioni > Tutorials > Ell	bow ~ C	Search Elbow		P
Save As	Ctil+s	Organize 🔻 New f	older					≣ ▪	•
Import Archive Ansys Minerva Scripting		 Documents Pictures Music 	* * *	Name	^	Date modified No items match your search.	Туре	Size	
Export Report		🛃 Videos	*						
1 D:\ \ElbowFluent\ElbowFl 2 D:\ \ElbowFluentSimulatio 3 D:\ \microChannelHeatSin 4 D: Woble \Laureand\Sara R	uentSimulation\ElbowFluent.wbpj n\ElbowSeptember2024.wbpj kFluent18.wbpj ossi Mel\ProvePerSaraRossiMel.wbpj	Tile name: File name: Save as type: Pro	1 powFluent pject Files (*.wbpj)]					~
Exit	C#I+Q								
Hydrodynamic Diffradion Hydrodynamic Response		∧ Hide Folders					Save	Cancel	

Figure 7: Saving and naming the project.



Figure 8: Order of generation of the elbow geometric elements.

Figure 8 shows the adopted order of generation of the elements.

• Right-click on the *Geometry* cell and select *New DesignModeler Geometry* from the menu, as shown in figure 9. Please note that double-clicking directly on the *Geometry* cell, corresponds to select a *New SpaceClaim Geometry*, i.e. the default geometry system. DM will start and you will need to select the unit of measure of lengths, as



Figure 9: Starting ANSYS Design Modeler.

shown in figure 10. Choose *mm* (selecting, in the checkbox, *Always use project units*, will always use the unit of measure of length previously chosen in WB).



Figure 10: Choice of length measurement unit in ANSYS DM.

• Select the *YZ* plane in the left tree and, with the right mouse button, select the *Look at* option to move the view on the plane itself. Then click on the *New Sketch* icon (**) located at the top, and rename the sketch (right

Then click on the *New Sketch* icon (15), located at the top, and rename the sketch (right mouse button) to *Diameter*. The DM window will appear as in figure 11.

- Move to the *Sketching* menu on the left and select *Circle* in the *Draw* submenu, as shown in figure 12. At this point:
 - Position the mouse pointer at the origin of the axes until the letter 'P' appears, indicating the option to *snap* to the point;



Figure 11: Generation of the sketch Diameter.



Figure 12: Selecting the *Circle* item in the *Draw* submenu in *Skeching*.

- Press the left mouse button, drag it radially and finally release it to define the radius;
- Select the *Dimensions* submenu to parametrically define the diameter value;
- Select the *Diameter* option;
- Move the mouse over the circumference;
- Click with the mouse to associate the *D1* dimension to the geometric entity;
- In the *Details* menu on the left, type the diameter value, 50 mm, in the appropriate box.

The final result is shown in figure 13.



Figure 13: Diameter selection.

- In the *Modeling* tree, select the sketch to extrude (*Diameter*), and:
 - Click on the *Extrude* icon **R** at the top;
 - Rename the extrusion to *UpstreamPipe*, similarly to what was done with the sketch;
 - Set the extrusion direction to *Reversed*;
 - Set the extrusion depth (*Depth*) to 200 mm;
 - Click on the *Generate* icon \neq , present in the top menu.

The result is illustrated in figure 14, in which the choices indicated are highlighted.

• Return to the YZ plane again and:



Figure 14: Generation of the pipe section upstream of the elbow.

- Generate a new sketch, to be renamed to Axis;
- Select the *Sketching* tools and from the *Draw* menu click on *Line*;
- Draw a horizontal line below the circumference, keeping the left mouse button pressed. To do this, make sure that there is a letter 'H' near the line, to indicate that the line is arranged horizontally: the result will be as in figure 15;
- Go to the *Dimensions* menu and select a *vertical* dimension;
- Click on the line to be quoted, just drawn, and on the horizontal axis of the circumference the Y axis to define the dimension;
- Define, as previously, the correct value of the quota, equal to 125 mm, as illustrated in figure 16.
- Select the *Revolve* modeling option, indicated by the symbol **\$\mathbf{n}\$**.
 - Rename the feature to *Bend*;
 - In the detail window, at the bottom left, associate the *Diameter* sketch to *Geometry*, and the *Axis* sketch to *Axis* (after selecting the *Axis* sketch with the mouse, simply press *Apply* to associate it with the rotation axis);
 - Set, always in the detail window, the revolution angle to 90°;
 - Click on \neq Generate: the result will be as in the figure 17.
- Rotate the pipe so as to bring the surface downstream of the elbow to the foreground, as depicted in figure 18:



Figure 15: Horizontal axis for the sketch Axis.



Figure 16: Quotation of the horizontal axis.



Figure 17: Generation of the elbow.

- Click on the **★** *New plane* icon;
- Rename the plane to *DownstreamPlane*;
- In the detail window, select, among the possible types, From Face;
- Move to *Base face*, and select the face in the viewpoint (surface downstream of the elbow) and then press *Apply*;
- Finally click on $\not \ge$ *Generate* to create the plane.
- Once the plane is generated, you need to:
 - Generate a new *Extrude* feature, renaming it *DownstreamPipe*, similarly to what was done previously with *UpstreamPipe*;
 - In the detail window, associate the newly created plane to *Geometry*: in this way, all the surfaces present in the plane are extruded;
 - Define an extrusion depth equal to 300 mm;
 - Click on \neq *Generate* to generate the feature.

The result is illustrated in figure 19.

At this point, is useful to set - or check - that, clicking the *part* to be named *pipe* in the Tree Outline on the left, is set as *fluid* in the *Details View* below the Tree Outline, as illustrated in figure 20.

• While not strictly necessary in our case, we save the geometry just completed with the name *ElbowGeometry*: the extension *.agdb*, Design Modeler DataBase files, will be assigned automatically, as shown in figure 21. Close *Design Modeler*.



Figure 18: Generation of the plane downstream of the elbow.



Figure 19: Generation of the pipe downstream of the bend.

Tree Outline	L	ą						
🖃 🧹 🚱 A: Elbo	w							
	🗸 🖈 XYPlane							
	ZXPlane							
🗄 🗸 🛧 Y	⊥ YZPlane							
🗄 🗸 🖪 U	UpstreamPipe							
🗄 🗸 👘 Be	end							
	ownstreamPlane							
🕂	ownstreamPipe							
ė ~®_1	Part 1 Body							
· · · · ·	🗊 Pipe							
Sketching Mod	feling							
Sketching Moo Details View	Jeling	ф						
Sketching Mod Details View	deling	ą						
Sketching Mod Details View Details of Body Body	leling y Pipe	Ą						
Sketching Moo Details View Details of Body Body Volume	Pipe 1.3673e+06 mm ³	ą						
Sketching Moo Details View Details of Body Body Volume Surface Area	teling Pipe 1.3673e+06 mm ³ 1.1331e+05 mm ²	4						
Sketching Moo Details View Details of Body Body Volume Surface Area Faces	Pipe 1.3673e+06 mm ³ 1.1331e+05 mm ² 5	4						
Sketching Moo Details View Details of Body Body Volume Surface Area Faces Edges	Pipe 1.3673e+06 mm ³ 1.1331e+05 mm ² 5 4	4						
Sketching Moo Details View Details of Body Body Volume Surface Area Faces Edges Vertices	teling Pipe 1.3673e+06 mm ³ 1.1331e+05 mm ² 5 4 0							
Sketching Moo Details View Details of Body Volume Surface Area Faces Edges Vertices Fluid/Solid	y Pipe 1.3673e+06 mm ² 1.1331e+05 mm ² 5 4 0 0 Fluid	₽						
Sketching Moc Details View Details of Body Body Volume Surface Area Faces Edges Vertices Fluid/Solid Shared Topo	teling Pipe 1.3673e+06 mm ³ 1.1331e+05 mm ² 5 4 0 Fluid Automatic	₽						
Sketching Moc Details View Details of Body Body Volume Surface Area Faces Edges Vertices Flud/Solid Shared Topo Geometry Type	teling Pipe 1.3673e+06 mm ² 1.1331e+05 mm ² 5 4 0 Fluid Automatic DesignModeler	₽						

Figure 20: Setting the pipe as fluid domain.

	5	8			
💽 Salva con n	ome				×
Salva in:	user_files	•	← 🗈 💣 📰-		
*	Nome	^	Ultima modifica	Тіро	
Accesso rapido Desktop Raccote Questo PC					
	<			_	>
	Nome file:	ElbowGeometry		•	Salva
	Salva come:	DesignModeler Database (*.agdb)		<u> </u>	Annulla

Figure 21: Saving the geometry.

• In the main project window (Workbench), save and update the entire project.

3.2 Generation of the computational grid

• In the main window, position the mouse on *Mesh* in the project scheme by right-clicking and selecting *Edit*, as in figure 22, or double-click with the left button.



Figure 22: Launching ANSYS Mesh.

• Once ANSYS Meshing has started, the result will be as in figure 23.

By clicking on *Mesh*, in the tree on the left of the project, the information reported in the detail window on the left, reproduced in figure 24, will be displayed. In particular, you will notice that *CFD* is indicated as *Physics Preference*, while *Fluent* is selected by default as *Solver Preference*, without the need for further information.

• It is now appropriate, before proceeding with the actual generation of the grid, to appropriately name all the boundary surfaces of the domain - *patches* - thus facilitating the subsequent imposition of the boundary conditions. As shown in figure 25, right-click on *Model* → *Insert* → *Named Selection* in the top left tree.

Next:

- Rename to *Inlet*;
- Select the entry face to the pipe upstream of the bend, recognizable by its position with respect to the coordinate axes (for convenience: the shortest branch);
- Press Apply, as shown in figure 26.

Proceed in a similar manner for the exit section (face), naming it Outlet, as in figure 27.

It is finally necessary to define the pipe wall, renaming the region to *Wall*, as illustrated in figure 28, paying attention to the fact that the wall is actually made up of 3 surfaces: to choose multiple surfaces you need to hold down the *Ctrl* key during the selection.

17



Figure 23: ANSYS Mesh splash screen.

D	etails of "Mesh"	
-	Display	
	Display Style	Use Geometry
=	Defaults	
	Physics Prefer	CFD
	Solver Prefere	Fluent
1	Element Size	Default (2,861
	Export Format	Standard
	Export Previe	No
+	Sizing	-
+	Quality	
+	Inflation	
+	Advanced	
-	Ctatictics	





Figure 25: Inserting Named Selection into ANSYS Mesh.

	\$		Con	text	B : Prove Elbow - N	Meshing [CFD P	repPost, CFD	PrepPo	st Pro, CF	D Base]						-		×
	File	Home	Named S	election	Display Sel	ection Aut	tomation	Learnir	ng and Su	pport	Motio	n		uick Laun	ch		^ [J 🕜
1	Duplicate	Cut Copy Paste Outline	× Delete Q Find ₽ <mark>c</mark> a Tree *	Generate Mesh	Named Select	tion 🖲 Imag ystem 🛱 Secti Secti Insert	es▼ on Plane otation	Tools L	ayout									
C	utline		····· ▼ ₽ ⊑	1×	ତ୍ର 🗐 🗑 🗑	🖏 🖺 🔿 '	- 💠 🍳 🤆	Q Q	(🤤 S	elect 🔫	Mode∗	F. 🗗		1 🖬 🕅		K.Y.Z 100		2 ¥
The second se	Name Project	t odel (B3) Geome Geome Geome Geome Geome Materia Materia Materia Materia Materia Materia Named	try Imports try Imports try als nate Systems d Selections nlet	, s	Selection 10/10/2024 22:42											A r	15 ` 202	YS 4R1
D	etails of "Ir	nlet" 👓	···· ▼ ₽ □	×			N											
	Scope Scoping N Geometry Definition Send to So Protected Visible	Aeth G A N Olver Ye Pr Ye	eometry Sele pply Cance															
-	Program C Statistics	Contr Ex	kclude				0,000)				0,3	3 00 (m))				X
	Туре	M	lanual - Colortin	•						0,150			,			1	ź	
F	leady				0	No Messages	1 Face Sel	ected: A	rea = 1,95	35e-003	m² 🔺	Metric (m, kg,	N, s, V, A)	Degree	s rad/s	Cels	ius 🖃

Figure 26: Definition of the Inlet face in ANSYS Mesh.



Figure 27: Definition of the Outlet face in ANSYS Mesh.

	6 🗄 ≏	Contex	t	B: Prove Elbow	- Meshing	[CFD PrepPost	, CFD Prep	Post Pro,	CFD Base]						-		1	×
	File Home	Named Sele	ection	Display S	election	Automatio	n Lea	rning and	Support	Motio	on		Quick La	unch		^	\square	0-
	Duplicate Understand	× Delete Q Find ₽₽ Tree ▼ G	enerate Mesh	Named Sele	ection (d System (Insert	D Images ▼ Section Plan Annotation	e Tools	Layout										
(Outline	···· + ₽ □ >	•	ତ୍ ତ୍ 📦	8 - E	0 - 💠 🤇	2 오 🍭	۵ ک	Select	≮ Mode≁	F. 6				×.Y.Z 100		 	•
	Name Project* Project* Name Nodel (83 Project* Project* Project* Name Project* Name Name Name Name Name Name Name Name		Si 11	election 1/10/2024 22:46				0		+					٨	ns 202	Y 4 F	ร ข
E	Scope		~															
E	Scoping Meth C Geometry / Definition Send to Solver Y Protected P Visible Y Program Contr E Statistics	eometry Sele Apply Cancel es trogram Contr es xclude					0,000				0	,300 (i	m)		Z	×	Υ	
	Type N	lanual							0,150									
	Total Salact	la Salaction	•		-0				0.400									
E.	Ready				U No	Messages	3 Faces Se	ected: Ar	ea = 0.109	12 m* 4	Metric	(m. k	a. N. s. V.	A) Dear	ees rad	/s Ce	sius	1

Figure 28: Definition of the Wall faces in ANSYS Meshing.

- It is useful to observe that Fluent recognizes typical naming conventions on boundaries, such as *inlet*, *outlet*, *far-field*, *symmetry*, *wall*, *interior*, or *internal*. More precisely, the changing of Fluent boundary types based on name patterns is controlled by an advanced option, which is enabled by default. When enabled, the following most common patterns and corresponding zone types are applied on all zones:
 - inlet maps to a velocity-inlet zone
 - outlet maps to a pressure-outlet zone
 - *symmetry* maps to a symmetry zone
 - far*field maps to a pressure-far-field zone
 - mass*inlet maps to a mass-flow-inlet zone
 - press*inlet maps to a pressure-inlet zone
 - *mass*outlet* maps to a mass-flow-outlet zone
 - outflow maps to an outflow zone
 - *inlet*vent* maps to an inlet-vent zone
 - *outlet*vent* maps to an outlet-vent zone
 - intake*fan maps to an intake-fan zone
 - *exh*fan* maps to an exhaust-fan zone
 - *fan* maps to a fan zone
 - *porous*jump* maps to a porous-jump zone
 - radiator maps to a radiator zone
 - overset maps to an overset zone
- Once the definition of the regions is complete, we can proceed to the generation of the meshes by adopting the *default* settings for what concerns the information on the minimum and maximum dimensions of the cells, the grid generation algorithm etc. This information can be found by clicking on *Mesh* → *Default* and *Sizing* in the tree on the left, as shown in figure 29.

In order to generate the mesh, you need to click on the *Generate Mesh* icon $(\mathcal{F}_{Generate})$ available under the *Home* tab in the main menu, or the $(\mathcal{F}_{Generate})$ available under the *Mesh* tab, and wait the time necessary to complete the operation.

The progress of the grid generation will be indicated by the *waitbar* present in the lower left corner of ANSYS Mesh and shown in figure 30. In this case since, as we will see, it is a rather coarse mesh, the time needed is negligible, while in the case of complex geometries and high resolution, the time needed can be of the order of hours.

The resulting grid, which will be displayed in the main window at the end of the generation, is shown in figure 31.

The grid thus obtained, obviously, is too coarse to adequately describe the phenomenon: it is sufficient to observe, by clicking on *Statistics* in the bottom left menu, that the total

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Size	5,e-003 m
Export Format	Standard
Export Preview Surface Mes	n No
Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1,2)
Max Size	5,e-003 m
Mesh Defeaturing	Yes
Defeature Size	Default (2,5e-005 m)
Capture Curvature	Yes
Curvature Min Size	Default (5, e-005 m)
Curvature Normal Angle	Default (18,°)
Capture Proximity	No
Bounding Box Diagonal	0,57228 m
Average Surface Area	2,2621e-002 m ²
Minimum Edge Length	0,15708 m
Quality	
Inflation	
Advanced	

Figure 29: Default values for sizing in ANSYS Mesh.

1	📕 (5%) Meshing

Figure 30: Grid generation progress in ANSYS meshing.

Ki 🖂 🗸 👘 🗸 🗛 Karat Shawat Shawa Mar	king (/ED DansDark /ED DansDark Dar /ED Para)	
Context E: Copy of Elbow - Mes	ining [CFD Preprost, CFD Preprost, Pro, CFD Base]	
File Home Mesh Display Selection	Automation Learning and Support Motion	Quick Launch 🔨 🐼 😯
Mesh Workflows* Mesh Workflow Mesh Workflow	Source/Target Method Sizing Face Method Sizing Metho	
Outline 👻 🖡 🗆 🗙	🔅 Q. Q. 📦 🖕 😂 😘 🔿 💘 Q. Q. Q. Q. Q. Select 🍡 Moder 📰 🕞 🕞 🐻 🐻 🐻 🐘 🌾 🕾 🖂 🗖 Clipboard - [Empty]	Extend *
Name Search Outline 🗙 –		•
Improved: Improved:		Ansys 2024 R1
Details of "Mesh"		
E Display		
Display Style Use Geometry Setting		
Defaults		
Physics Prefere CFD		
Solver Prefere Fluent		
Element Size Default (2,8614e-002 m)		
Export Format Standard		
Export Preview No		
Sizing		
Quality		
Inflation		
Advanced		
Statistics		
Nodes 4482		
Elements 3526		
Show Detailed No		Y Y
		· •
		7
		\sim
	0,000 0,100 0,200 (m)	· · · · · · · · · · · · · · · · · · ·
	0.050 0.152	-
	0,000 0,100	
Ready	Messages pane No Selection 🔺 Metric (m	n, kg, N, s, V, A) Degrees rad/s Celsius 🔬

Figure 31: Grid obtained, in ANSYS Mesh, using the *default* values for the Sizing.

22

number of nodes is equal to 4482, and the total number of cells (remember that ANSYS Fluent is a *Cell-Centered Finite Volume* type code), is equal to 3526.

Therefore, it is necessary to modify (reduce) the default values for *Sizing*, appropriately reducing the size of the cells. For example, by bringing both the values of *Element Size*, which sets the global maximum mesh size on surfaces, and *Max Size*, which sets the global maximum mesh size within volumes, to 5 mm (5×10^{-3} m), as illustrated in figure 32, the mesh of figure 33 is obtained.



Figure 32: Setting reduced values for Sizing in ANSYS Mesh.



Figure 33: Grid obtained, in ANSYS Mesh, using the values of Sizing of figure 32.

• It is possible to note, however, that also this grid is probably inadequate, especially with regard to the wall resolution. We will return to this aspect later. It is worth remembering that, every time changes are made, it is necessary to *Update* the mesh, in addition to

updating any references to the mesh by other cells, in our case the *Setup* cell. This is done either by clicking *Update* (\bigcup_{update}), instead of *Generate Mesh* ($\bigcup_{Generate}$) in DM, or by right-clicking the *Mesh* cell in WB and selecting *Update*, as shown in figure 34



Figure 34: Update of the Mesh cell in WB.

Once the mesh generation is complete, we can now close ANSYS meshing (from File \rightarrow Close meshing) and proceed with the Setup of the problem, but not before saving the entire project in WB (File \rightarrow Save).

3.3 Problem setup

• In the main window, position the mouse, in the WB project scheme, on the *Setup* cell by clicking with the right button and select *Edit*, as in figure 35.

Once the Setup is started, the *Fluent Launcher* window will appear, as depicted in figure 36. In this window, select *Double Precision* and a number of *Solver Processes* equal to 1 or 2.³

When the launch of ANSYS Setup will be completed, the result will be as shown in figure 37.

- It is now necessary to proceed with the setting of all the parameters and modes concerning the simulation, and in particular:
 - Analysis type (steady or non-steady) and solver type (pressure-based or density-based);
 - Presence of rigid body motions (dynamic mesh);
 - Characteristics of the computation domain;

³It is important to remember that the number of solver processes has to be always less than the number of the physical cores of the machine where Fluent runs.

Perturement Solver of the state of th		•	E ⁸ eluidelau (el			
Image: Data System S		1	Geometry	ient)		
Protet Laurcher 2004 R1 (Setting Laurch) Protet Restong Protecker Protecker Protecker Protecker Protecker Protecker <		3	Mesh			
Pertret Laurcher Pertret Per		4	Setup		_	
Pretr Laurcher Pretr Pretres Pretr Laurcher Pretr Pretres Pretr Pretr Pretres		5 🧃	Solution	Edit		
Eber Import Fuent Case And Data Import Fuent Case <		6 🧕	Results	Register Startup Scheme File		
Import Runch Case Import Runch Case			Elbov	Import Fluent Case And Data	•	
Impart ROM Impart Rom <th></th> <th></th> <th></th> <th>Import Fluent Case</th> <th>•</th> <th></th>				Import Fluent Case	•	
Public Lipstream Components Care Generated Data Care General Options Care General Options <tr< th=""><th></th><th></th><th></th><th>Import ROM</th><th>•</th><th></th></tr<>				Import ROM	•	
Present Launcher 2024 R1 (Setting Edd Only) Present La				Duplicate		
Prent Launcher Prent Launcher Prent Launcher None Prent Launcher None Prent Launcher None None Solver Options Ob outbit precision One Prent Launcher None Dime Renaine Dime Prent Launcher None Dime Dime Prent Launcher Dime <				Transfer Data From New	•	
Update Updat				Transfer Data To New	•	
Implete Upstreen Components Reet Reet Reet Quark Help Add Note Figure 35: Starting ANSYS Setup. Filent Launcher 2024 RI (Setting Edit Only) Fuent La			7	Update		
Clear Generated Data Refresh Resen Resen Properties Quick Help Ad Note				Update Upstream Components		
Properties Quick Help Add Note Figure 35: Starting ANSYS Setup. Fleent Launcher 2024 R1 (Setting Edit Only) Solver Options Option 1 Solver Options Option Solver Options Option Solver Options Option Solver Options Option Solver Processes Solver Processes Z Parallel (Local Machine) Solver Processes Z Processes Z				Clear Generated Data		
Rest Poperties Quick Help Add Note Figure 35: Starting ANSYS Setup. Figure 42: Starting Editory Fount Launcher 2024 R1 (Setting Edit Only) Fuent Launcher 2024 R1 (Setting Edit Only)			¢	Refresh		
Properties Quick Help Add Note Figure 35: Starting ANSYS Setup. Figure 35: Starting ANSYS Setup. Figure 1 Seture			_	Reset		
Properties Quick Help Ad Note Figure 35: Starting ANSYS Setup. Figure 35: Starting ANSYS Setup. Prent Launcher 2024 R1 (Setting Edit Only) Fuent Launcher 2024 R1 (Setting Edit Only) One Ceneral Options Parallel Settings Remote Scheduler Environment One Solver Options O Do not show this panel again Parallel (Local Machine) Solver Processes 2 Verking Directory Cytrogram Files/MSYS Incly241/Framework/bin/Win64			ab	Rename		
Quick Help Add Note Figure 3.5: Starting ANSYS Setup. Figure 3.5: Starting ANSYS Setup. Figure 3.5: Starting ANSYS Setup. Figure 2.2: Control of the setup setu				Properties		
Add Note Figure 35: Starting ANSYS Setup. Fuent Launcher 2024 RI (Setting Edit Only) Fuent Launcher Fuent Launcher Fuent Launcher Image Ceneral Options Parallel Settings Solver Options Image Parallel (Local Machine) Solver Processes Image Vorking Directory				Quick Help		
Figure 35: Starting ANSYS Setup. Please Launcher 2024 R1 (Setting Edit Only) Fuent Launcher 2024 R1 (Setting Edit Only) Fuent Launcher Fuent Launcher Immession On the General Options Parallel Settings Remote Scheduler Environment Dimension 20 30 30 30 30 30 30 30 31 32 33 33 34 30 31 32 33 34 35 36 37 38 39 39 30 30 31 32 33 34 34 35 36 37 38 39 39 39 39 30 30 31 32 33 34 35 36 37 38 39 <t< th=""><th></th><th></th><th></th><th>Add Note</th><th></th><th></th></t<>				Add Note		
Fluent Launcher 2024 R1 (Setting Edit Only) Fluent Launcher General Options Parallel Settings Remote Scheduler Environmer Dimension 2D O Do not show this panel again Parallel (Local Machine) Solver Processes 2 Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64						
Home General Options Parallel Settings Remote Scheduler Environmer Dimension 2D	🥌 Fluent Launcher 2	024 R1 (Setting Edit Only)	/			– 🗆 X
Home General Options Parallel Settings Remote Scheduler Environmer Dimension 2D © Double Precision © Double Precision 0 Do not show this panel again Parallel (Local Machine) Solver Processes 2 • • • • Solver Processes 2 • • • • • • Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64 • • • • •	Fluent Lau	Incher				/\nsys
Dimension Solver Options ○ 20 ○ Double Precision ③ 30 ○ Do not show this panel again Parallel (Local Machine) 2 Solver Processes 2 Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64	Home	General Options	Parallel Set	ings Remote	Scheduler	Environment
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64	Dimension			Solver Options		
	○ 2D			Double Precision		
Parallel (Local Machine) Solver Processes 2 • Vorking Directory C:\Frogram Files\ANSYS Inc\v241\Framework\bin\Win64 •) 3D			Do not show this	panel again	
Solver Processes	Parallel (Local I	Machine)				
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64	Solver Processes			2		
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64						
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64						
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64						
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64						
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64						
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64						
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64						
Vorking Directory C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64						
C:\Program Files\ANSYS Inc\v241\Framework\bin\Win64	Working Directory					
Start Cancel Help 🚽	C:\Program Files\A	NSYS Inc\v241\Framewo	rk\bin\Win64			•

Figure 36: Fluent Launcher window.



Figure 37: ANSYS Fluent Setup initial screen.

- Presence of sub-domains;
- Thermophysical properties of the materials;
- Boundary conditions;
- Interfaces (overset meshes);
- Solver parameters and schemes;
- Initialization;
- Output options;
- Grid adaptivity;
- Execution parameters.

In the graphical user interface (GUI) of ANSYS Fluent, the *ribbon*, located at the top of the Fluent GUI, is the primary method for setting up and running the simulation. It facilitates access to the most commonly used items, with tabs nominally arranged in a *left to right* workflow for a typical simulation. Contents within each tab are grouped with related content and are organized logically to accommodate varied projects and priorities. There is the option to minimize the ribbon by clicking (**•**), located to the right of the ribbon tabs.

It will therefore be useful and convenient to follow this order to prepare the launch of the simulation, with the warning that it is not necessary to set the irrelevant items, which should therefore be left at their default values (e.g. in this case there are no rigid body motions, nor sub-domains, nor interfaces, and furthermore we do not activate the adaptivity of the grid). • In order to display the mesh lines in the graphic window, there are two options: you either click the *Display* button located in the *Mesh* tab of the ribbon (Declay...), or the analogous button located in the toolbar of the graphics window. In both cases, a window like in figure 38 will show up, and it will be necessary to tickmark the *Edges* checkbox. The result will be as shown in figure 39.

Mesh Displa	ау		×
Options Nodes Geges Faces Partitions Overset Shrink Factor 0 Transparency	Edge Type All Feature Outline Feature Angle 20 0	Surfaces Filter Text	×
Outline I Adjacency	Interior	New Surface 🖕	
		Colors Close Help	

Figure 38: Set Fluent mesh display options.

Please note that, in order to maximize the graphic window, it is possible to collapse both the *Task Page* and the *Outline View*, located at the left of the graphic window, by clicking the *Collapse* buttons () located on top of both windows.

The first step to do is to check that the mesh is correct and that it corresponds to our system. In order to do so, click the Check → Perform Mesh Check button in the Domain tab of the ribbon. This action will provides a mesh check report, displayed in the console window located below the graphics display, with details about domain extents, statistics related to cell volume and face area, and information about any problems associated with the mesh. In our case the output should be like this

```
Domain Extents:
x-coordinate: min (m) = -2.000000e-01, max (m) = 1.500000e-01
y-coordinate: min (m) = -2.500000e-02, max (m) = 2.500000e-02
z-coordinate: min (m) = -4.250000e-01, max (m) = 2.500000e-02
Volume statistics:
minimum volume (m3): 5.408590e-09
maximum volume (m3): 1.384248e-07
total volume (m3): 1.358336e-03
Face area statistics:
minimum face area (m2): 1.081718e-06
maximum face area (m2): 3.842509e-05
Checking mesh......
Done.
```



Figure 39: Fluent mesh shown in graphics window

• A second check to do consists in a quick evaluation of the mesh quality. To do it, click on *Quality* → *Evaluate Mesh Quality* button in the Mesh tab of the ribbon, shown in figure 40.



Figure 40: Check Mesh Quality in Fluent

The result of this check is, in our case, is this

```
Mesh Quality:
Minimum Orthogonal Quality = 4.61947e-01 cell 14488 on zone 2 (ID: 29573
on partition: 0) at location ( 1.18191e-01, -5.68914e-03, -7.60437e-02)
Maximum Aspect Ratio = 1.26192e+01 cell 9705 on zone 2 (ID: 29551 on
partition: 0) at location ( 3.34879e-03, -5.68914e-03, 2.88517e-03)
```

It is useful to remind that a more comprehensive evaluation of the quality of the mesh

can be done with ANSYS Mesh, clicking on the *Metric Graph* button (Metric) located in the Mesh tab of the ribbon and selecting the mesh metric desired, i.e. *Aspect ratio, Skewness, Orthogonal quality* et.

• It is convenient, for our purposes, to use *mm* as length unit. To do so, click on the *Units* button (**Units..**) also located in the Domain tab of the ribbon. This will open a new window, where it will be possible to select the units of measurement for each physical quantity. In particular, for length we select *mm*, as shown in figure 41.

🥌 Set Units		×
Quantities	Units	Set All to
heat-flux-resolved heat-generation-rate heat-transfer-coefficient ignition-energy kinematic-viscosity length length-inverse length-time-inverse	m cm mm in ft	default si british cgs
mag-permeability mass mass-diffusivity	Factor 0.001 Offset 0	
	New List Close Help	

Figure 41: Set length unit in Fluent.

It is important to note that, when you read the mesh into Ansys Fluent, it is always assumed that the unit of length is *meters*.

- It is now necessary to proceed to the definition of the parameters of the problems, and in sequence:
 - 1. Select the *Physics* tab of the Fluent ribbon.
 - 2. Check that, in the *Task Page*, the solver selected is the *Pressure-Based* with *Absolute* Velocity Formulation and that the *Steady* formulation is chosen, as indicated in figure 42

Task Page			<			
General			?			
Mesh						
Scale	Ch	eck	Report Quality			
Display	Uni	ts				
Solver						
Туре		Velocit	v Formulation			
• Pressure-Ba	ised	 Absolute 				
🔾 Density-Bas	ed		elative			
Time						
 Steady 						
() Transient						
Gravity						

Figure 42: Solver selection in Fluent Task page.

3. Click *Models* in the *Setup* tree branch on the left, and check that in the Models window that will appear, all the options are set to *off*, but the *Viscous* model, that should be set to *SST k-omega*, as shown in figure 43. If this is not the case, clicking on *Viscous* under the Models tab of the ribbon, you will be able to select the appropriate turbulence model for our problem, as illustrated in figure 44. Please note that, since in this case there is not heat transfer, the *Energy equation* entry should be set to off.



Figure 43: Checking the active models in Fluent.

4. We have now to set up the material for our problem. We have to use the same values of the thermophysical properties reported in table 1. In the *Materials* group

of the *Physics* ribbon tab, click on *Create/Edit* ... icon(.....). In the *Create/Edit Materials* dialog box that will appear, type *water* for *Name* and in the *Properties* group enter the values of density and dynamic viscosity given in table 1. Click *Change/Create* and the result will be as in figure 45. A *Question* dialog box will open, asking if you want to overwrite *air*. Click *Yes*, in order to avoid any possible confusion later, e.g. only water will be used as a fluid (note that this will affect only the *local* copy of material properties).

- 5. In the *Zones* group of the *Physics* ribbon tab, click on *Cell Zones*. This will open the *Cell Zone Conditions* dialog box⁴. In the dialog box, only one zone should be visible, the one named *pipe*. Select it, and verify that in the *type* drop-down list it should be set to *fluid*. Click *Edit* to open the *Fluid* dialog box and Check that, on the *Material Name* drop-down list, *water* is selected. Click *Apply* to close the *Fluid* dialog box.
- 6. It is now necessary to setup the boundary conditions for the problem. In the same *Zones* group of the *Physics* ribbon tab, click on *Boundaries*. This will open the

⁴Please note that the same dialog box can be opened by double-clicking the *Cell Zone Conditions* under the *Setup* tree branch on the left.

Siscous Model	×
Model	Model Constants
	Alpha*_inf
Laminar	
🔿 Spalart-Allmaras (1 eqn)	Alpha inf
🔿 k-epsilon (2 eqn)	0.52
💿 k-omega (2 eqn)	
Transition k-kl-omega (3 eqn)	Beta*_Inf
Transition SST (4 eqn)	0.09
O Reynolds Stress (7 eqn)	a1
Scale-Adaptive Simulation (SAS)	0.31
O Detached Eddy Simulation (DES)	Beta_i (Inner)
 Large Eddy Simulation (LES) 	0.075
k-omega Model	Beta_i (Outer)
O Standard	0.0828
O GEKO	TVE (Toper) Prandtl #
O BSL	
⊙ sst	
⊖ WJ-BSL-EARSM	TKE (Outer) Prandtl #
k-omega Options	1
Low-Re Corrections	SDR (Inner) Prandtl #
	2
Near-Wall Treatment	SDR (Outer) Prandtl #
correlation	• 1.168
Options	Production Limiter Clip Factor
Curvature Correction	10
Corner Flow Correction	
Production Kato-Launder	
Production Limiter	
	User-Defined Functions
Transition Options	Turbulent Viscosity
Transition Model none	▼ none ▼
ок са	ncel Help

Figure 44: Set the viscous model in Fluent.

Create/Edit Materials				×
Name		Material Type	0	order Materials by
water		fluid	•	 Name
Chemical Formula		Fluent Fluid Materials		Chemical Formula
		water	•	
		Mixture		Fluent Database
		none		GRANTA MDS Database
				User-Defined Database
Pro	operties			
	Density [kg/m³] c	onstant 🗸 🗸	Edit	
	9	97		
	Viscosity [kg/(m s)] c	onstant 🔹	Edit	
		000000		
	Ľ	.0008899		
4				
		Change/Create Delete Close Help		

Figure 45: Inserting water properties in the Create/Edit Materials dialog box in Fluent.

Boundary Conditions task page⁵. It is convenient to display the boundary zones grouped by *Zone Type*: this can be achieved by clicking the *Toggle Tree View* button in the upper right corner of the *Boundary Conditions* task page, and under the *Group By* select *Zone Type* from the drop-down menu, as shown in figure 46.

oundary Conditions		?	
one Filter Text	5		
- Inlet		List View	
inlet		Group By	
interior-pipe		Name	
- Outlet		d Zone Type	
outlet		♥ Zone type	
- Wall			
wall			¢*

Figure 46: Boundary conditions grouped by Zone Type in Fluent.

(a) Set the boundary conditions at the (fluid) *inlet*.

From the *Zone* selection list, select *inlet*, for the *Type* select *velocity-inlet* and click *Edit*. As illustrated in figure 47, in the *Velocity Inlet* dialog box, set the *Velocity Magnitude* equal to 0.2 *m/s*, and leave all other parameters for the *Momentum* tab as the default. Click *Apply* to close the *Velocity Inlet* dialog box.

🥌 Velo	ocity Inlet							\times
Zone Nar inlet	ne							Î
Moment	um Thermal						UDS	
Velo	city Specificat	ion Method	Magnitude	e, Norm	al to Bounda	ary		•
	Refere	ence Frame	Absolute					•
	Velocity	/ Magnitude	[m/s] 0.2	2				•
Superso	nic/Initial Gau	ge Pressure	[Pa] 0					-
1	urbulence							
	Specificatio	on Method I	intensity a	ind Visc	osity Ratio			-
	Turbulen	t Intensity [[%] ₅					-
	Turbulent Visc	osity Ratio	10					-
•								
			Appl	y d	ose Help			



(b) For the *outlet*, select *outlet* from the *Zone* selection list, for the *Type* select *pressure-outlet* and click *Edit*. As illustrated in figure 48, in the *Pressure Outlet* dialog box, leave the *Gauge Pressure* equal to 0 *Pascal*, check the *Average Pressure Specification* and leave all other parameters for the *Momentum* tab as the default. Click *Apply* to close the *Pressure Outlet* dialog box.

⁵Please note that the same task page can be opened by double-clicking the *Boundary Conditions* under the *Setup* tree branch on the left.

Pressur	e Outlet						 	×
Zone Name								
outlet								
Momentum							UDS	
	Backflow	Reference f	rame Abs	olute				-
		Gauge Pre	essure [Pa]	0				
	Pressur	e Profile Mu	ltiplier 1					
Backflow Dir	ection Spe	cification M	ethod Nor	mal to	Boundary			-
Backflow Pressure Specification Total Pressure								
Preven	t Reverse	Flow						
🗌 Radial I	Equilibrium	Pressure Di	istribution					
🖌 Averag	e Pressure	e Specificati	on					
Target	Mass Flow	Rate						
Turbul	ence							
	Spec	ification Me	thod Inter	nsity a	nd Viscosity	Ratio		-
В	ackflow Tu	rbulent Inte	nsity [%]					-
Backflo	w Turbule	nt Viscosity I	Ratio 10					
			Ap	oply	Close	lelp		

Figure 48: Outlet boundary condition as pressure-outlet in Fluent.

(c) For the *wall*, select *wall* from the *Zone* selection list, for the *Type* select *wall* and click *Edit*. Verify that, in the *Momentum* tab, it is set as a *No Slip, Stationary Wall*, as depicted in figure 49.

3.4 Solution

Select the Solution ribbon tab and then:

- 1. Click on the Solution Methods icon (
- 2. This will open the *Solution Methods* task page: leave all the choices as their default values, like in figure 50.
- 3. In the *Reports* group, click the *Residuals* icon (MRESIDUAL). This will open the *Residual Monitors* dialog box and on this:
 - (a) Check that, in the Options group, Print to Console and Plot are enabled.
 - (b) Change the values for the Convergence Absolute Criteria from 0.001 to 1×10^{-4} for the continuity, the velocity components, turbulent kinetic energy and omega, as illustrated in figure 51.
- 4. It is a recommended practice to monitor physical quantities relevant to the problem at hand, in addition to equation residuals, when judging convergence. These physical quantities can be, for example, the *force* or *drag* for an *external aerodynamics* problem, or, like in this case, the *pressure loss* for an *internal flow* problem et. For this purpose:

33



Figure 49: Wall boundary condition in Fluent.



Figure 50: Solution methods selection in Fluent.

Residual Monitors				×	
Options	Equations				
Print to Console	Residual	Monitor Ch	eck Converg	ence Absolute Criteria	
Plot	continuity			1e-04	
Curves Axes	x-velocity			1e-04	
Iterations to Plot	y-velocity			1e-04	
1000	z-velocity			1e-04	
Iterations to Store				1e-04	
1000 🗘	omega			1e-04	
Convergence Conditions Show Advanced Options					
CK Plot Cancel Help					

Figure 51: Convergence criteria in the Residual Monitors dialog box in Fluent.

(a) We first define two surfaces, one just before the bend and one immediately after: select the *Results* ribbon tab and, in the *Surface* group, click *Create* \rightarrow *Plane...* and, in the corresponding window that will open up, name the surface as *up-streamplane*, as the Method select *YZ plane* and type 0 mm as *X* value. Click *Create*.

Operate in the same way for the *downstreamplane*, but in this case select XY Plane as Method and insert -125 mm as Z value.

Figure 52 illustrates the definition of the two planes.

Plane Surface ×	Plane Surface X
New Surface Name upstreamplane	New Surface Name downstreamplane
Method	Method
YZ Plane 🔻	XY Plane 💌
X [mm] 0 Select with Mouse Surfaces 1 \$ Reset	Z [mm] -125 Surfaces 1 Reset
Create Close Help	Create Close Help

Figure 52: Upstream and downstream surfaces definition in Fluent.