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 table 1. In the table, D is the diameter of the pipe, U_m is the mean velocity of the fluid and Re is the *Reynolds* number.

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	=	8.899×10^{-4} kg/(m s)
ρ	=	997 kg/m^3
Re	=	$\frac{\rho U_m D}{\mu}$ = 9980

Table 1: Data for the 90° elbow.

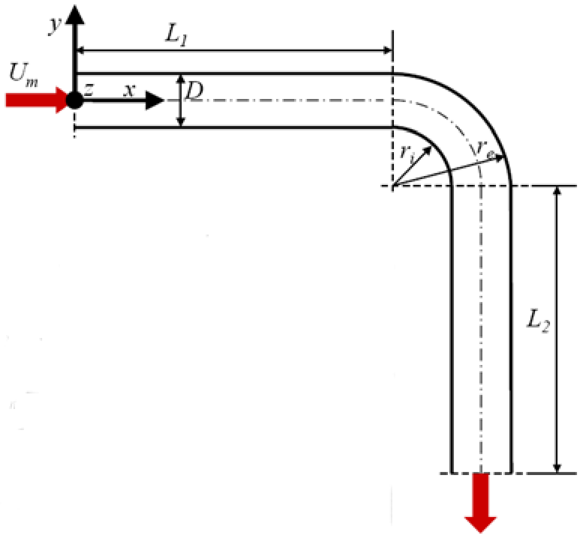


Figure 1: Characteristic dimensions of the 90° elbow.

In past experimental studies [1] 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.

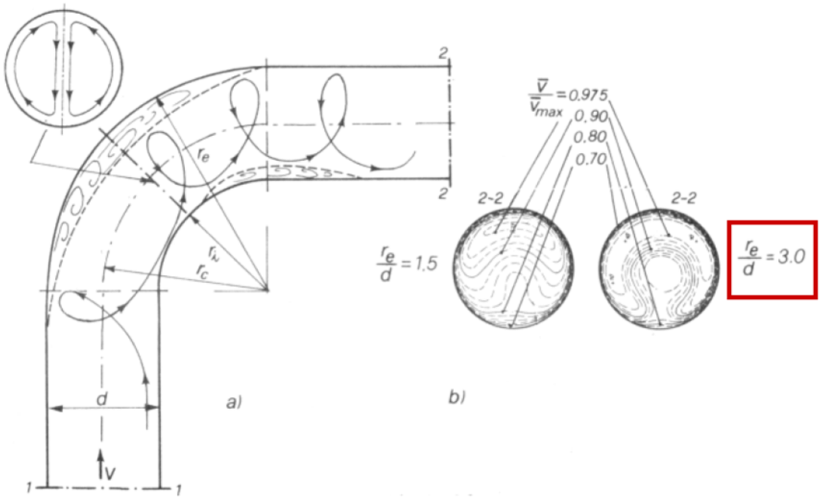


Figure 2: Characteristics of the flow in a 90° elbow detected experimentally.

3 Analysis with ANSYS Workbench and Fluent

- Start ANSYS Workbench (WB for brevity) via the icon on the desktop, or via

  Start → ANSYS 2024 R1 → Workbench 2024 R1

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

- 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.

- 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.

- 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*) 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.

  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 (✓).
  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:

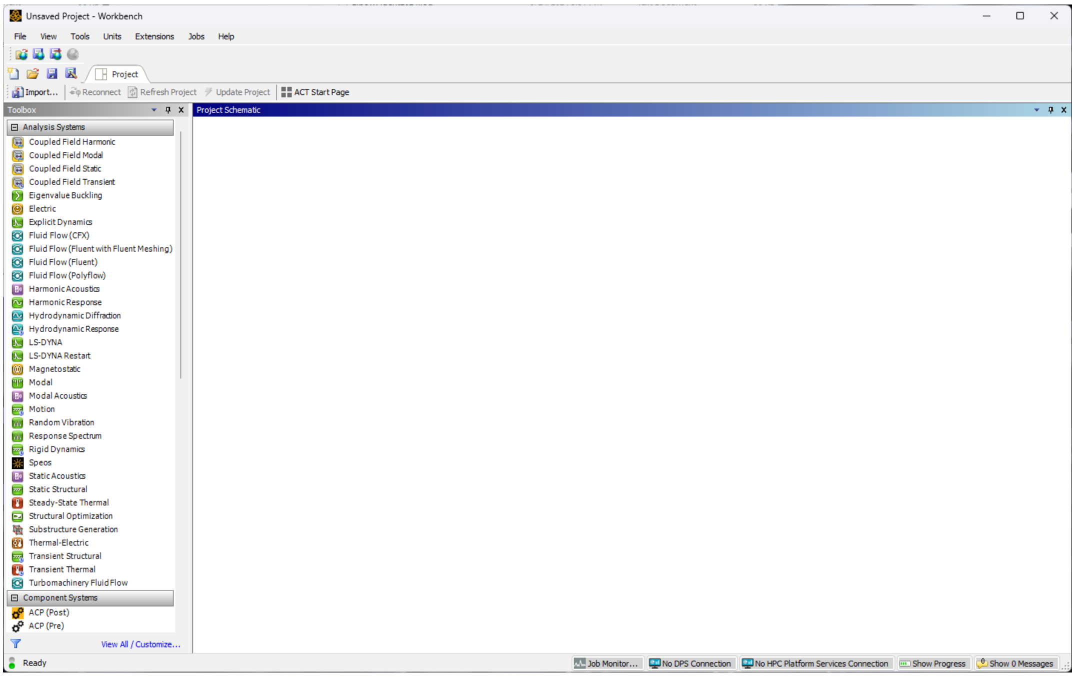


Figure 3: Starting ANSYS WB.

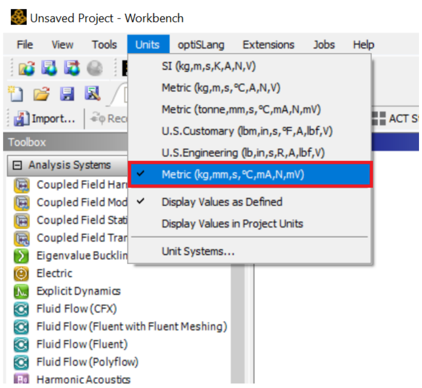


Figure 4: Selecting measurement units in ANSYS WB.

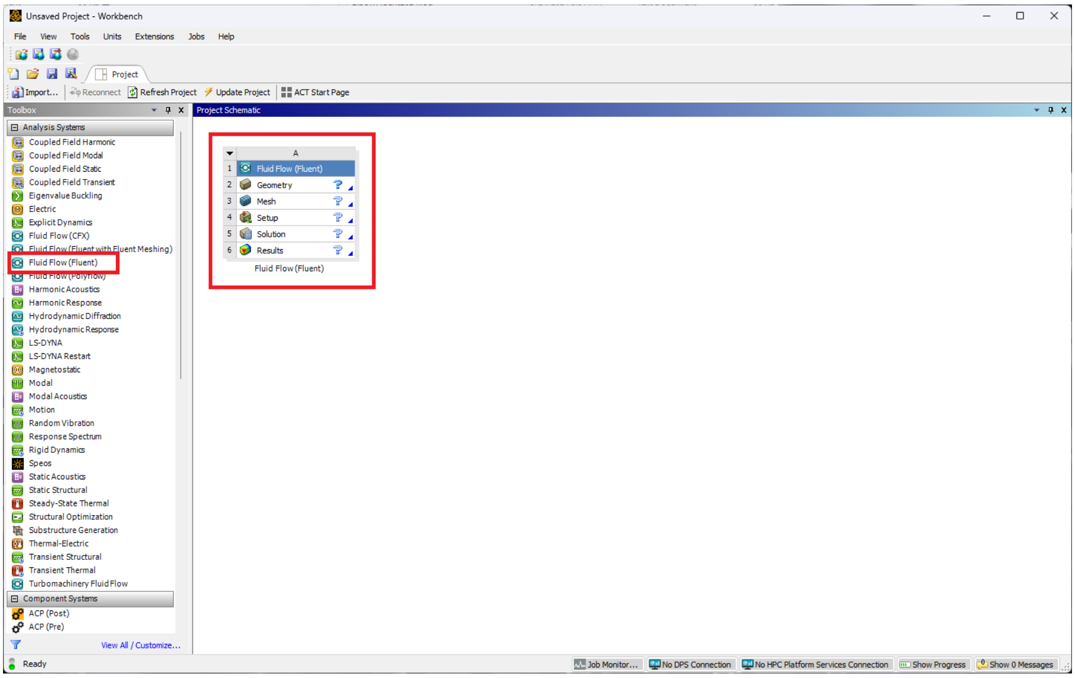


Figure 5: Selection of Fluent in ANSYS WB.

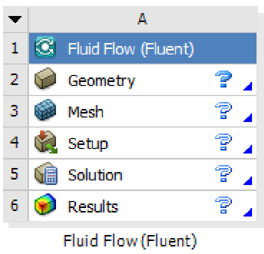


Figure 6: Cells status in the Fluent project.

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

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

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.

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.

[1] Selecting 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.

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 solution-specific 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.

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.

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 from the menu bar or from *File* $\rightarrow$ *Save as*, and name it *ElbowFluent* (the extension *.wbpj*, from *WorkBench ProJect*, will be assigned automatically), as in figure 7.

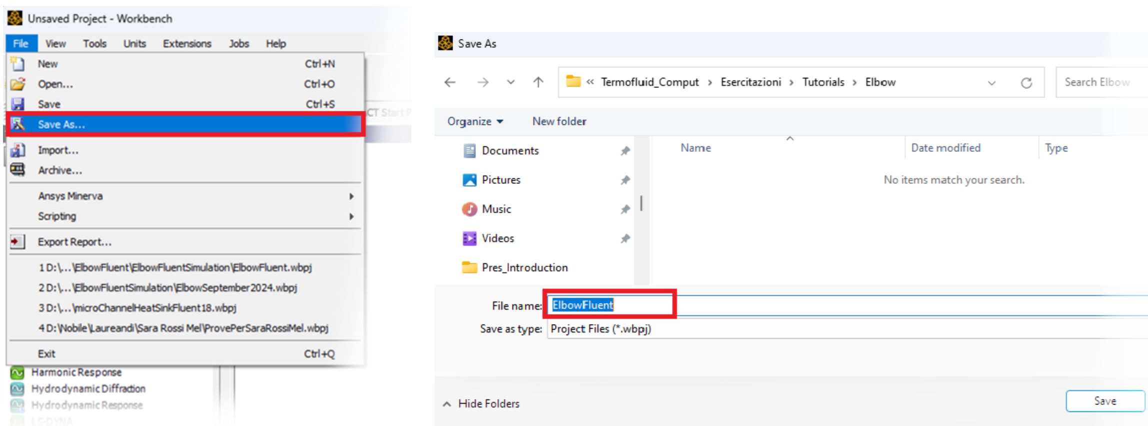


Figure 7: Saving and naming the project.

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).

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

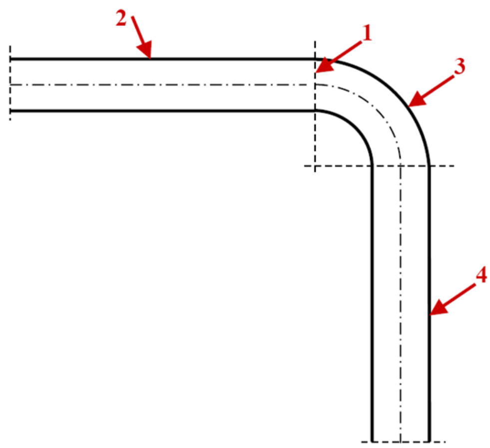


Figure 8: Order of generation of the elbow geometric elements.

• 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

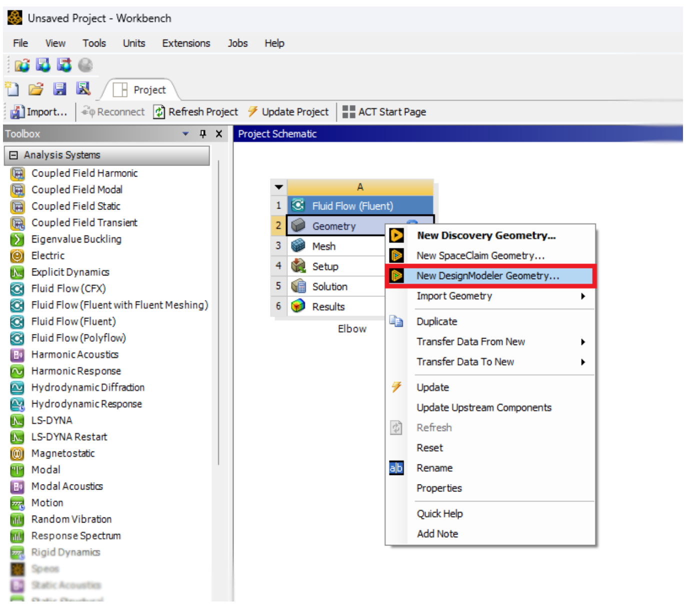


Figure 9: Starting ANSYS Design Modeler.

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).

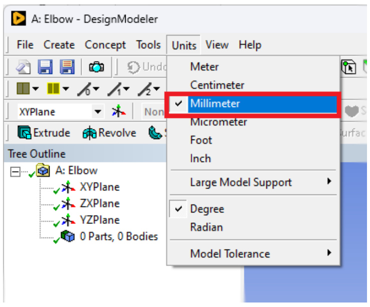


Figure 10: Choice of length measurement unit in ANSYS DM.

- 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 mouse button) to *Diameter*. The DM window will appear as in figure 11.

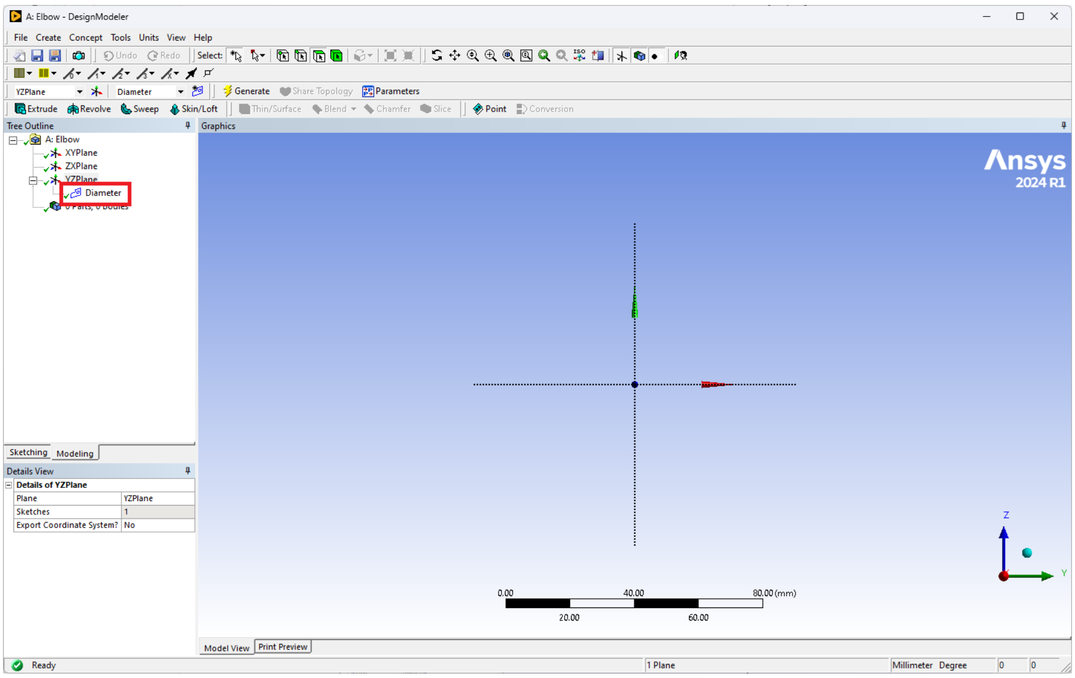


Figure 11: Generation of the sketch *Diameter*.

- 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;

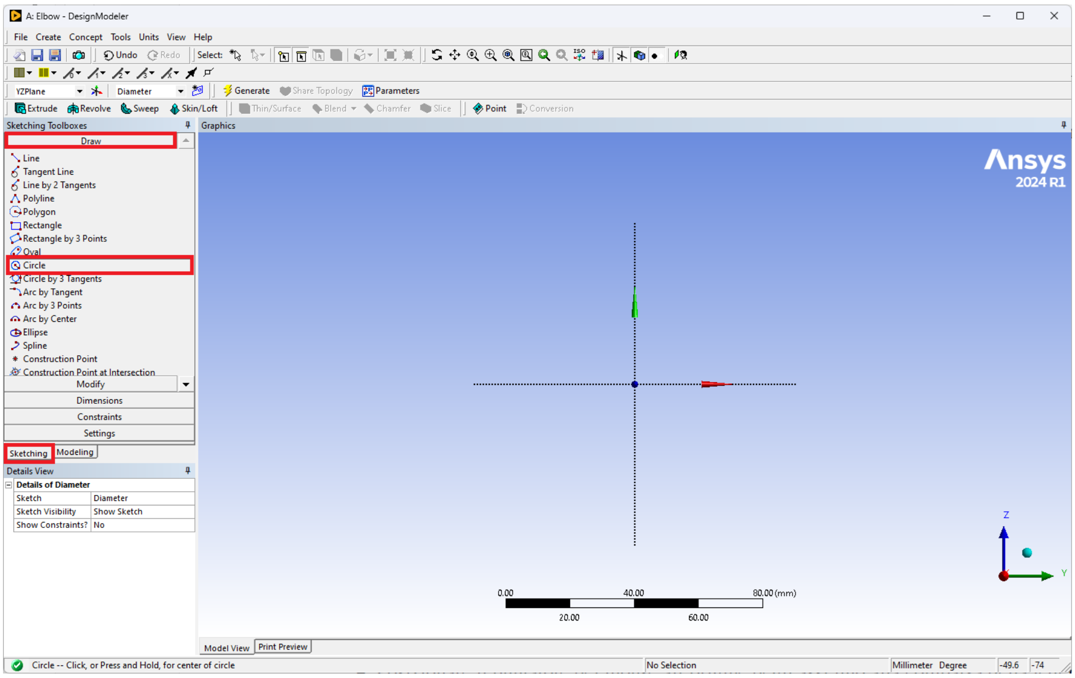


Figure 12: Selecting the *Circle* item in the *Draw* submenu in *Skeching*.

- 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.

- In the *Modeling* tree, select the sketch to extrude (*Diameter*), and:
  - Click on the *Extrude* icon 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, 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:
  - Generate a new sketch, to be renamed to *Axis*;

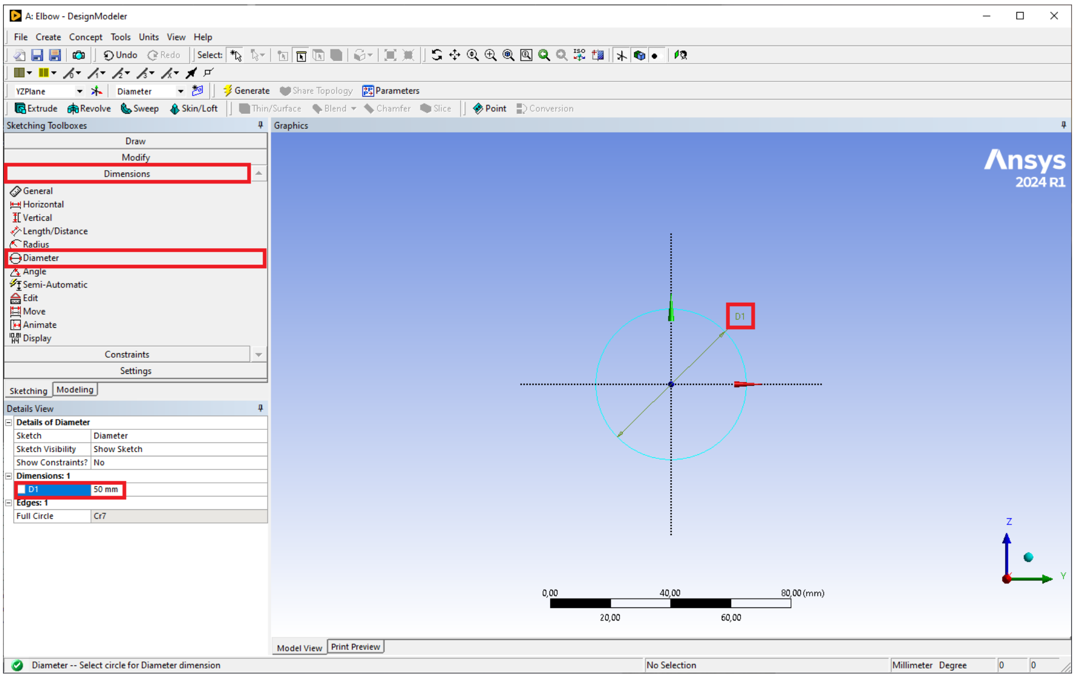


Figure 13: Diameter selection.

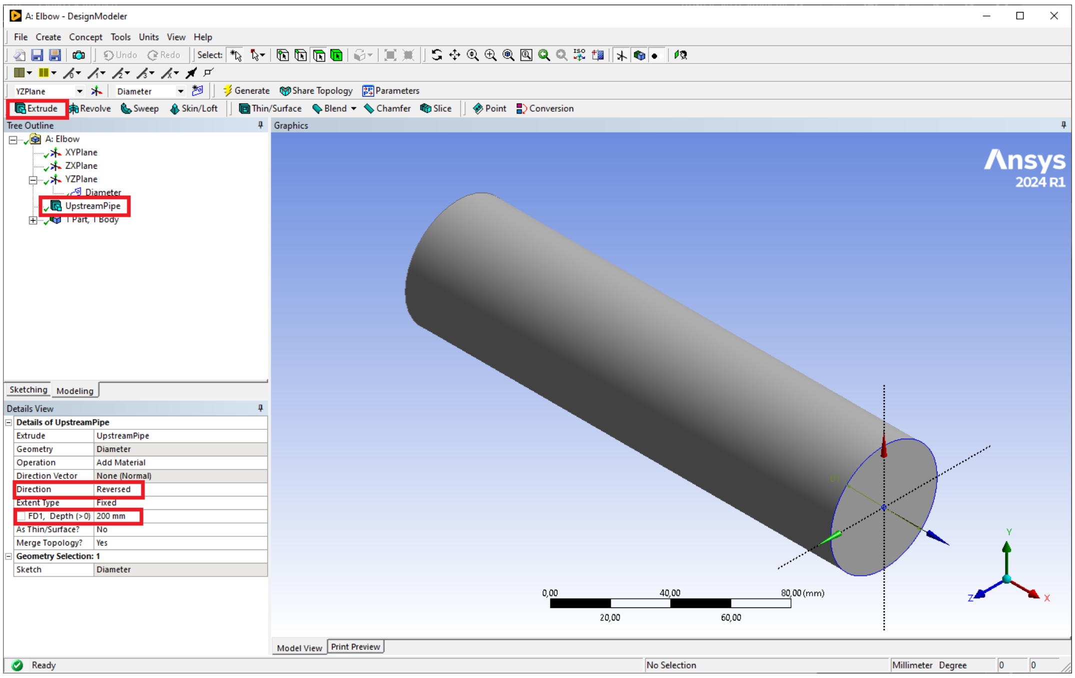


Figure 14: Generation of the pipe section upstream of the elbow.

- 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.

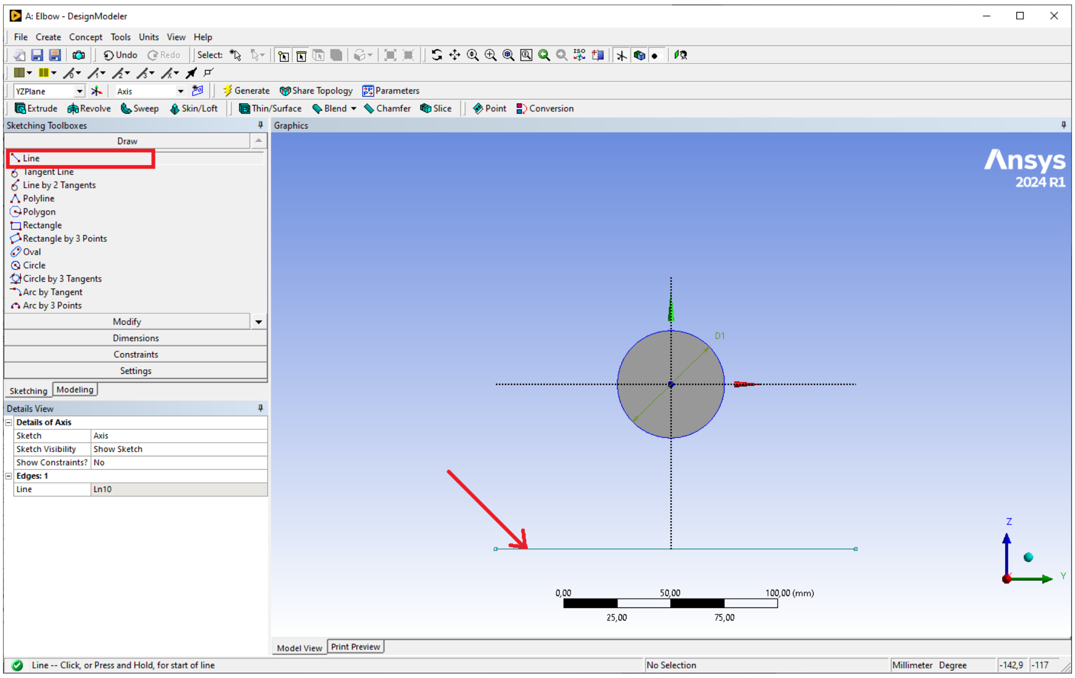


Figure 15: Horizontal axis for the sketch *Axis*.

- Select the *Revolve* modeling option, indicated by the symbol .
  - 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 *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, and select it, as depicted in figure 18.
- Once the face is selected, you need to:

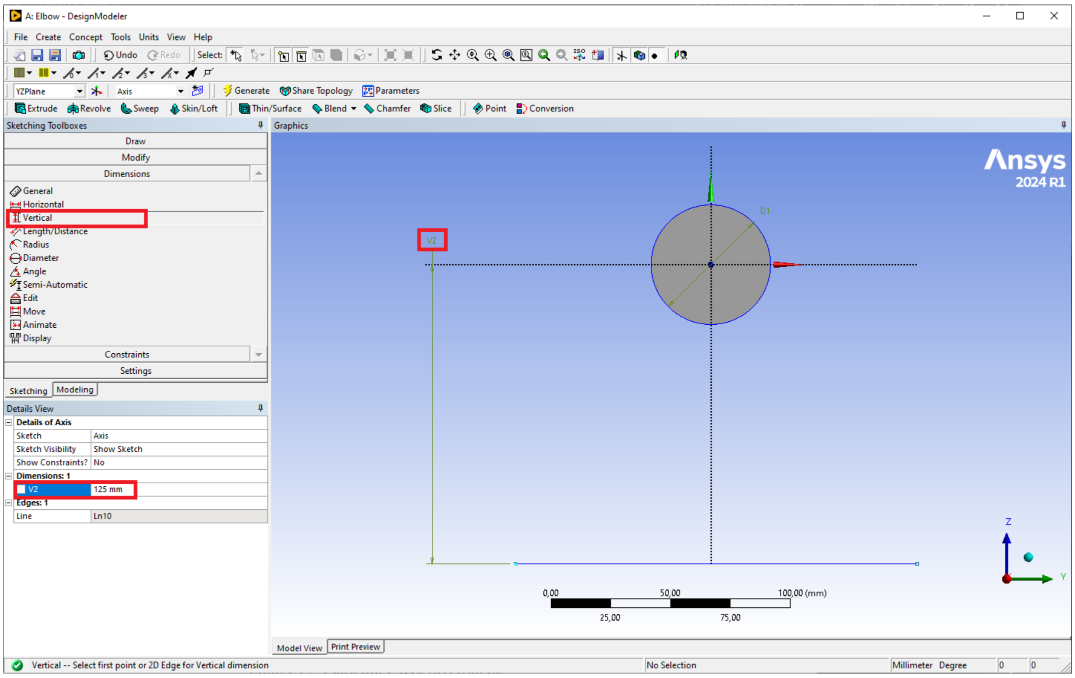


Figure 16: Quotation of the horizontal axis.

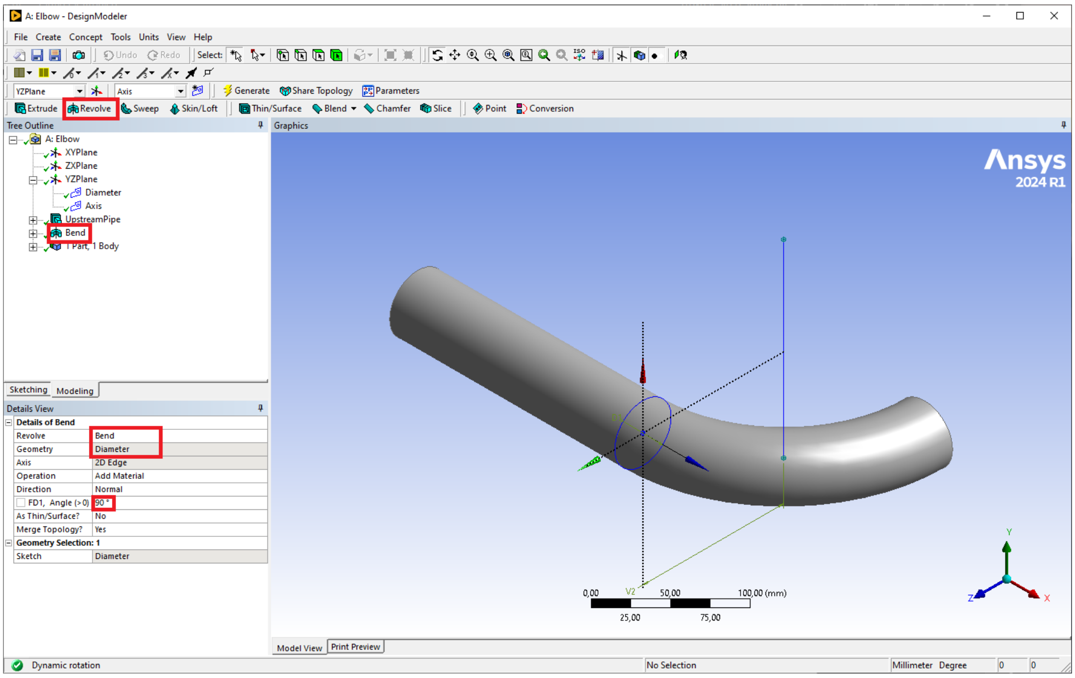


Figure 17: Generation of the elbow.

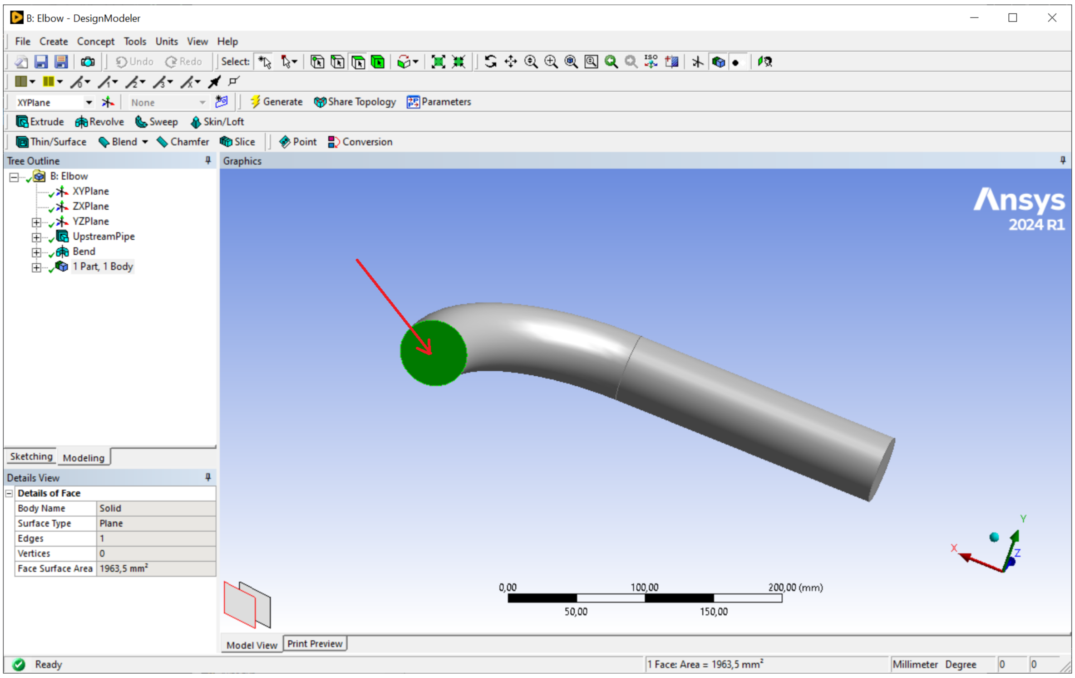


Figure 18: Selection of the face downstream of the elbow.

- Generate a new *Extrude* feature, renaming it *DownstreamPipe*, similarly to what was done previously with *UpstreamPipe*;
- In the detail window, associate the selected surface to *Geometry* and as direction vector select the XY Plane in the Tree Outline;
- Set the extrusion direction to *Reversed*;
- Define an extrusion depth equal to 300 mm;
- Click on *Generate* to generate the feature.

The result is illustrated in figure 19.

- In the *Modeling* tree on the left, click on *1 Part 1 Body*, and rename the corresponding solid as *pipe*, as shown in figure 20. Finally, is useful to set - or check - that, clicking on the *pipe* in the tree Outline, it is set as *fluid* in the *Details View* below, as also 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*.

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

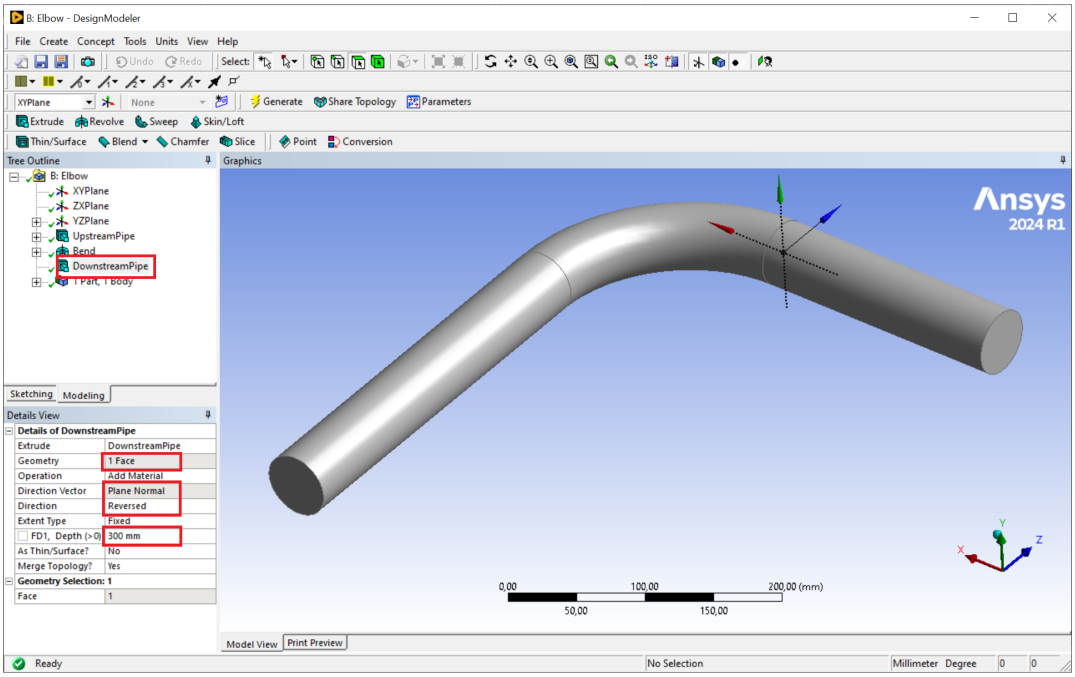


Figure 19: Generation of the pipe downstream the bend.

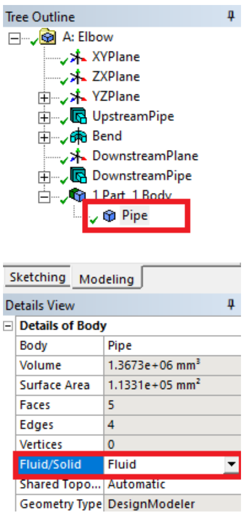


Figure 20: Renaming the solid as *pipe* and set it as a *fluid domain*.

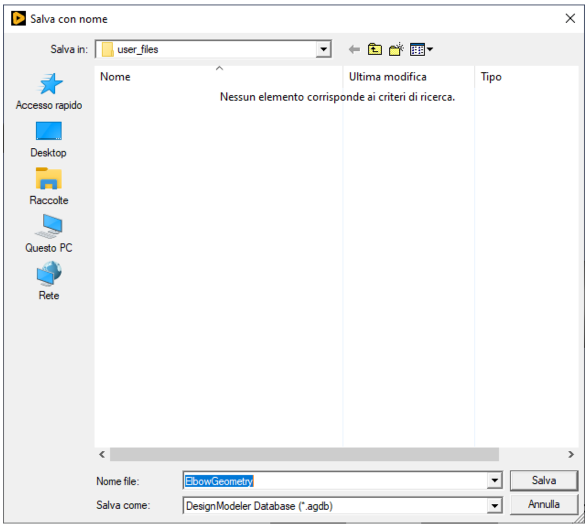


Figure 21: Saving the geometry.

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.

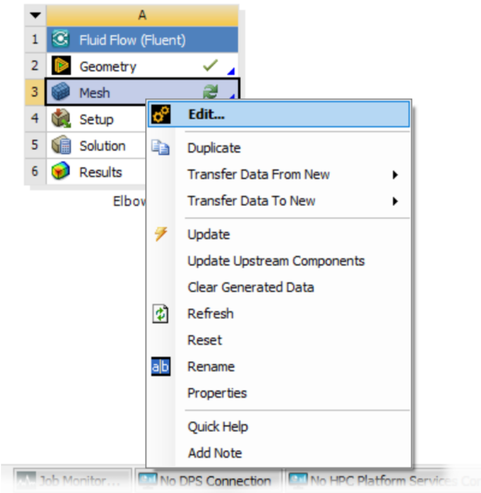


Figure 22: Launching ANSYS Mesh.

- 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.

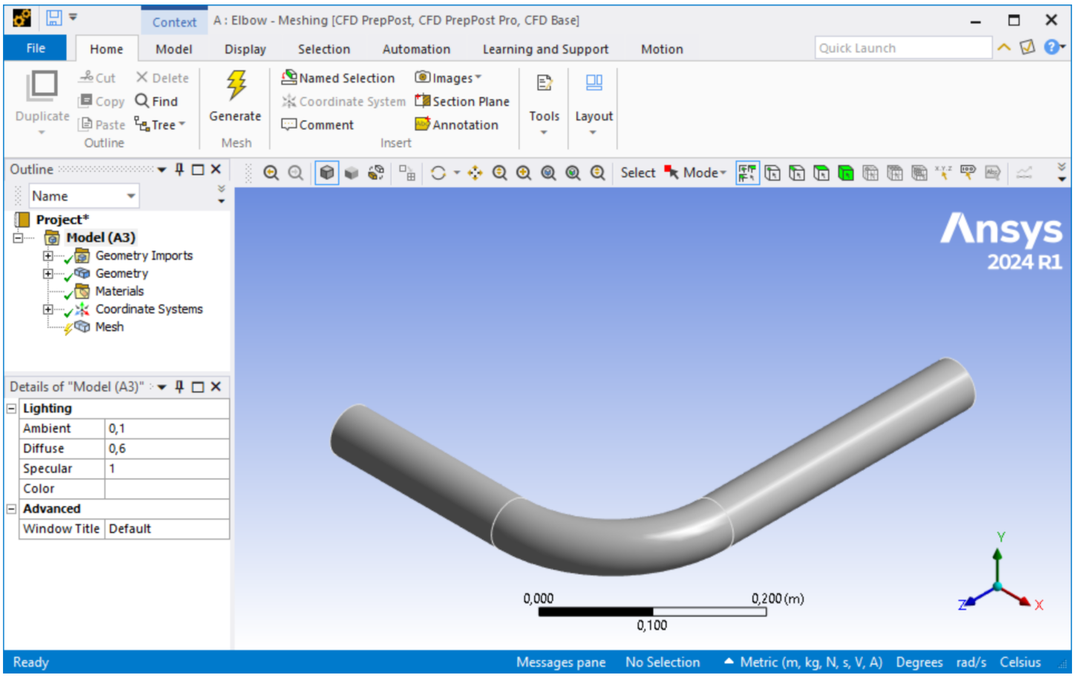


Figure 23: ANSYS Mesh splash screen.

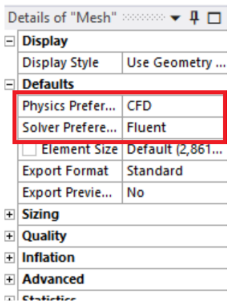


Figure 24: Details about *Preferences* in ANSYS Mesh.

- 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.

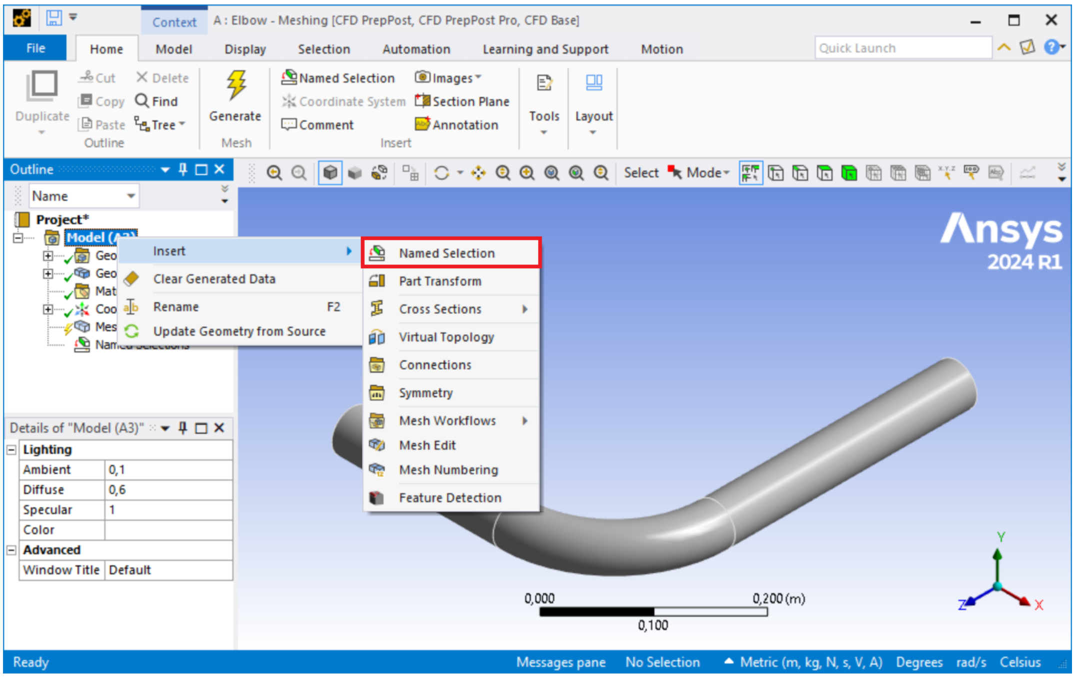


Figure 25: Inserting *Named Selection* into ANSYS Mesh.

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.

- 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

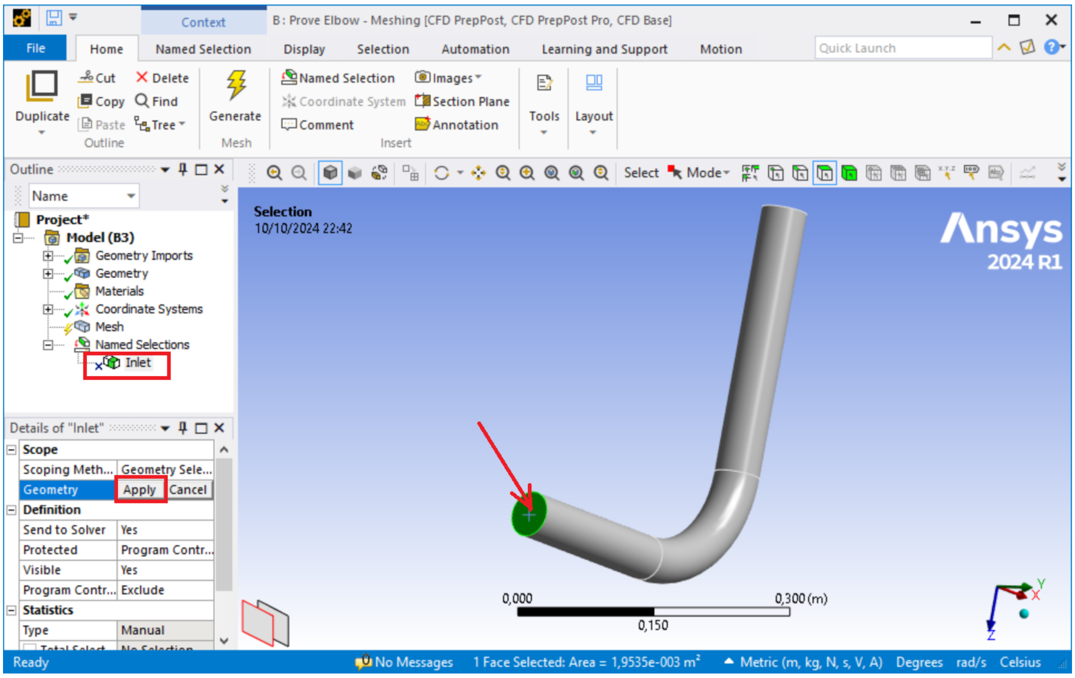


Figure 26: Definition of the *Inlet* face in ANSYS Mesh.

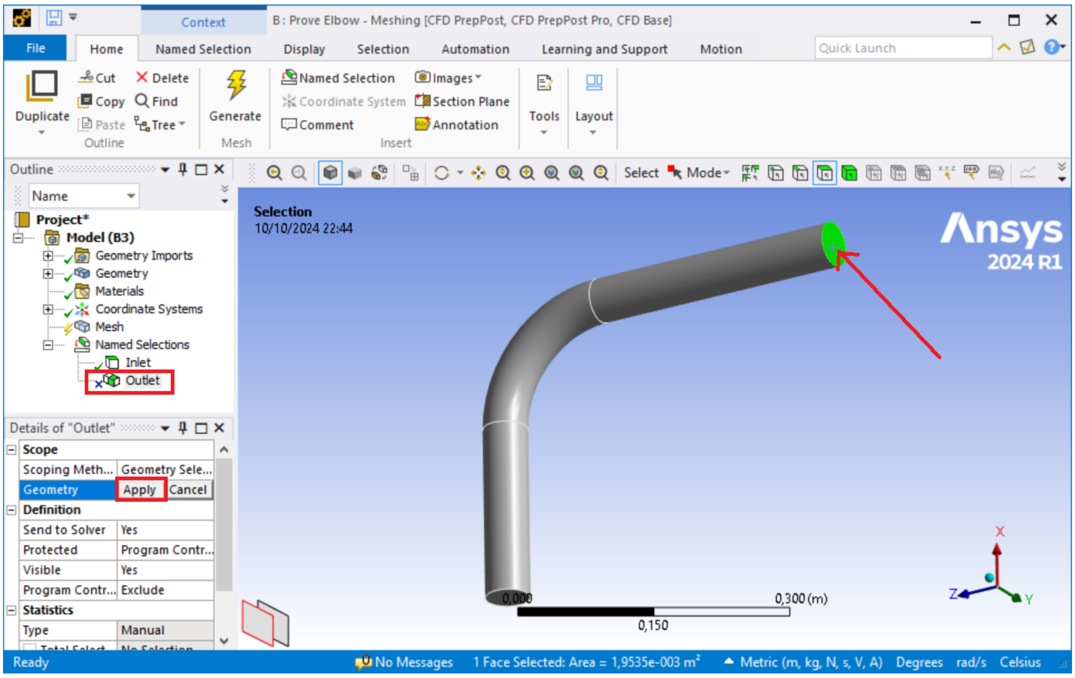


Figure 27: Definition of the *Outlet* face in ANSYS Mesh.

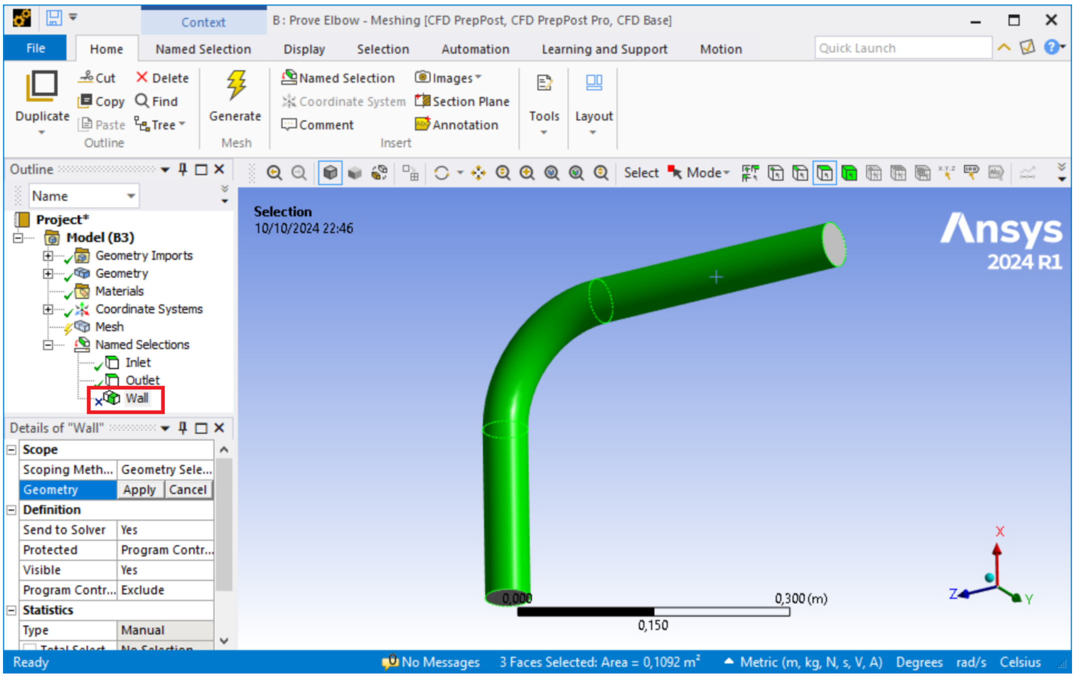


Figure 28: Definition of the *Wall* faces in ANSYS Meshing.

- *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.

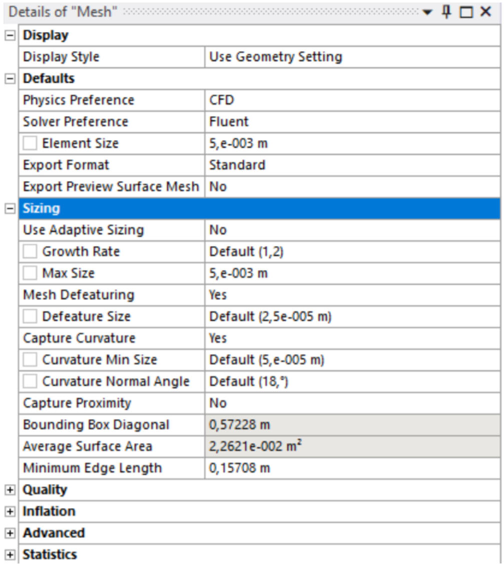


Figure 29: Default values for *sizing* in ANSYS Mesh.

In order to generate the mesh, you need to click on the *Generate Mesh* icon (Generate) available under the *Home* tab in the main menu, or the (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 rather small, while in the case of complex geometries and high resolution, the time needed can be of the order of hours, depending also on the computing platform.

Figure 30: Grid generation progress in ANSYS meshing.

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, clearly, 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 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.

- It is possible to note, however, that also this grid is probably inadequate, especially with

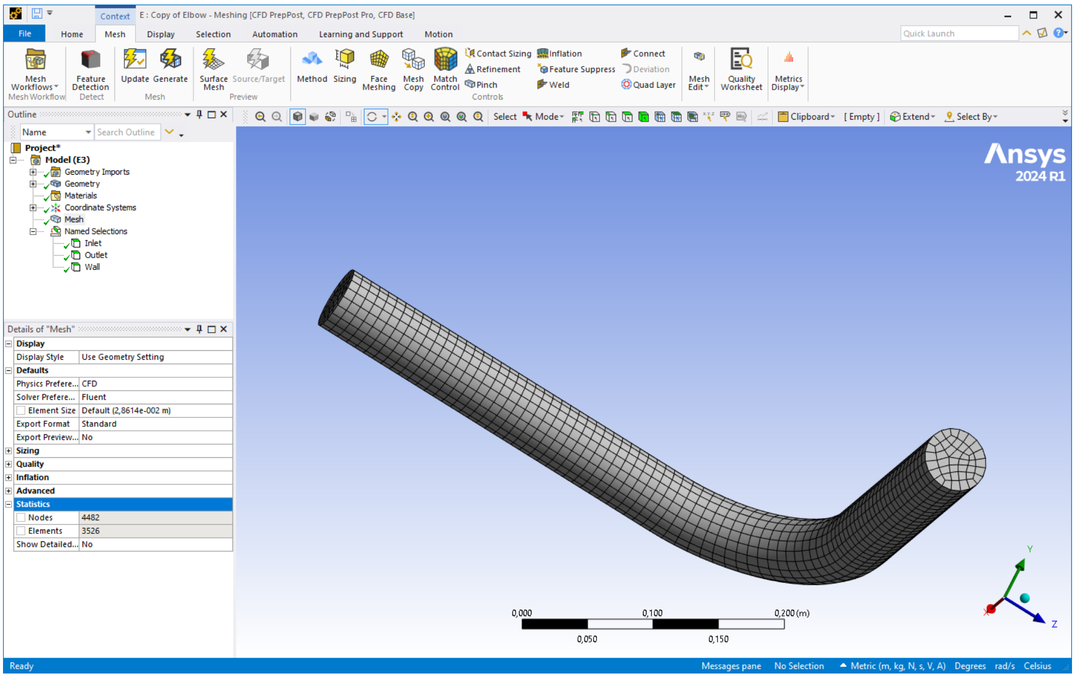


Figure 31: Grid obtained, in ANSYS Mesh, using the *default* values for the *Sizing*.

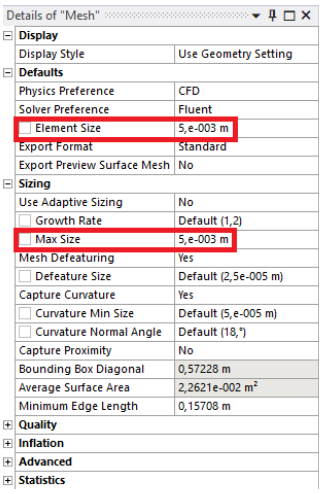


Figure 32: Setting reduced values for *Sizing* in ANSYS Mesh.

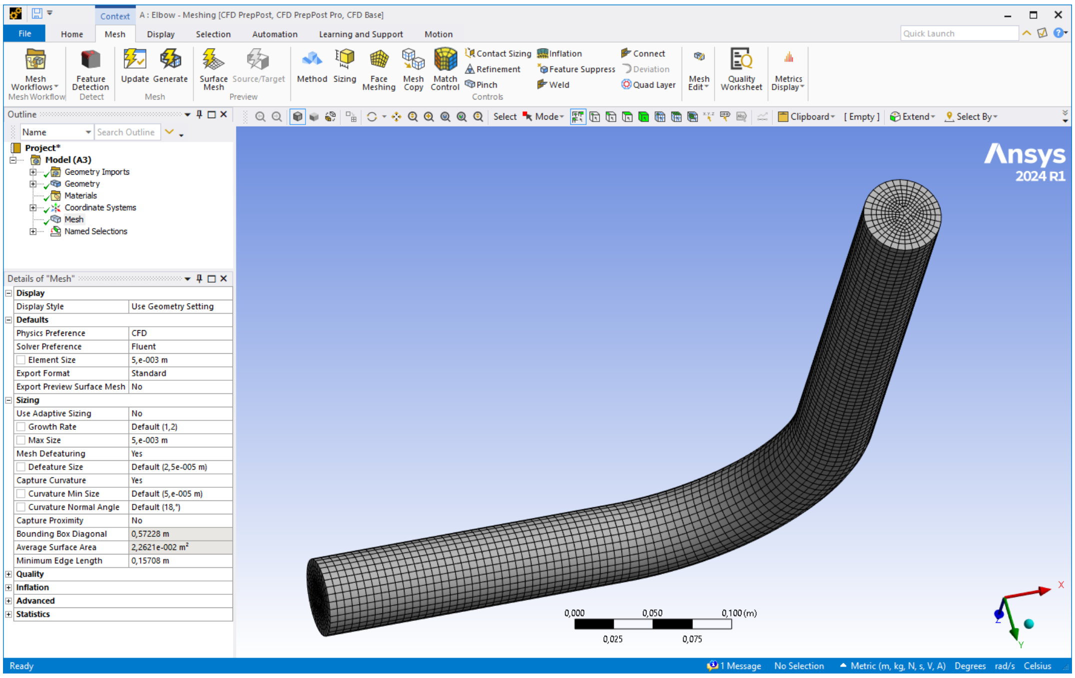
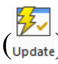


Figure 33: Grid obtained, in ANSYS Mesh, using the values of *Sizing* of figure 32.

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* (
), instead of *Generate Mesh* (
) in DM, or by right-clicking the *Mesh* cell in WB and selecting *Update*, as shown in figure 34

Once the mesh generation is complete, we can now close *ANSYS meshing* (from *File* → *Close meshing*) and proceed with the *Setup* of the problem, but not before saving the entire project in WB (*File* → *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.[2]

  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 models concerning the simulation, and in particular:

[2]It 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.

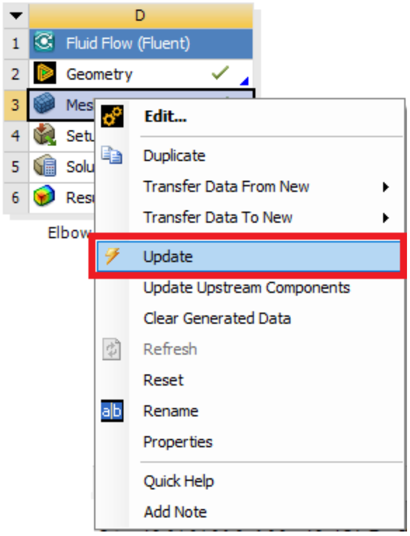


Figure 34: *Update* of the *Mesh* cell in WB.

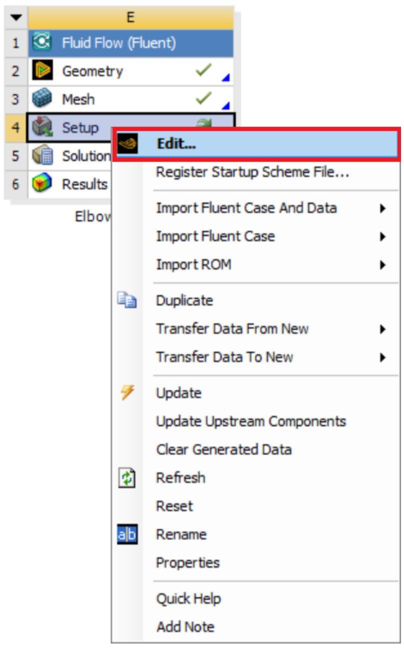


Figure 35: Starting ANSYS Setup.

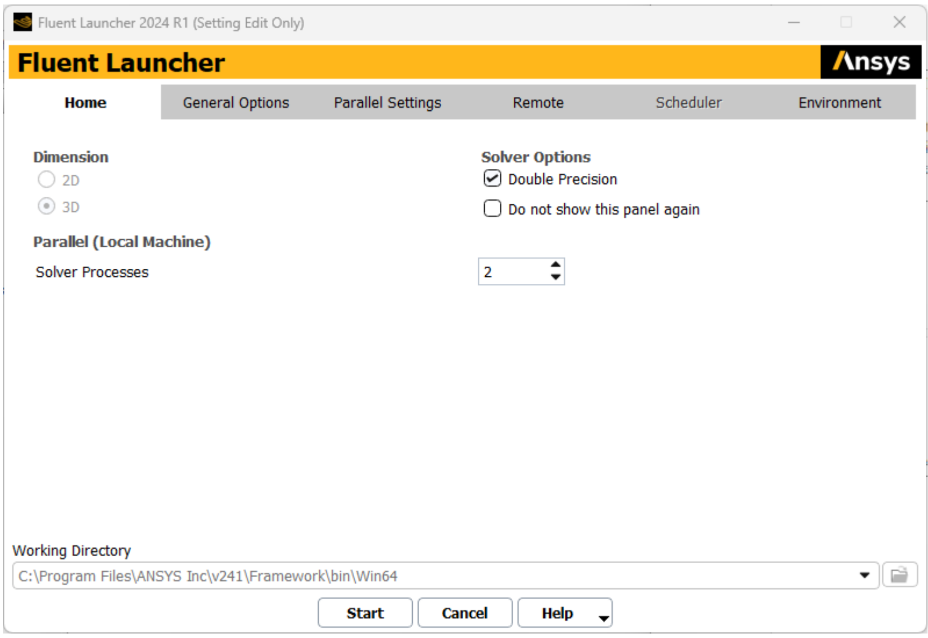


Figure 36: Fluent Launcher window.

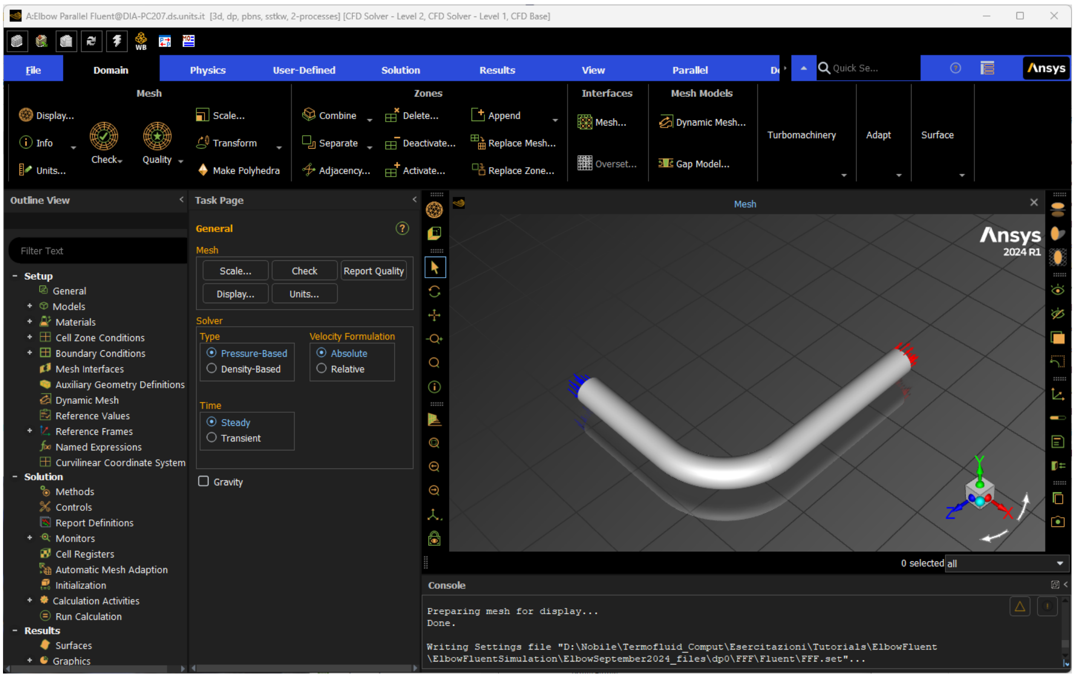


Figure 37: ANSYS Fluent Setup initial screen.

- 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;
- Presence of sub-domains;
- Thermophysical properties of the materials;
- Boundary conditions;
- Interfaces (overset meshes);
- Solver parameters and schemes;
- Initialization;
- Output and monitor 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, no sub-domains, no 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 (Display...), 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.

  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

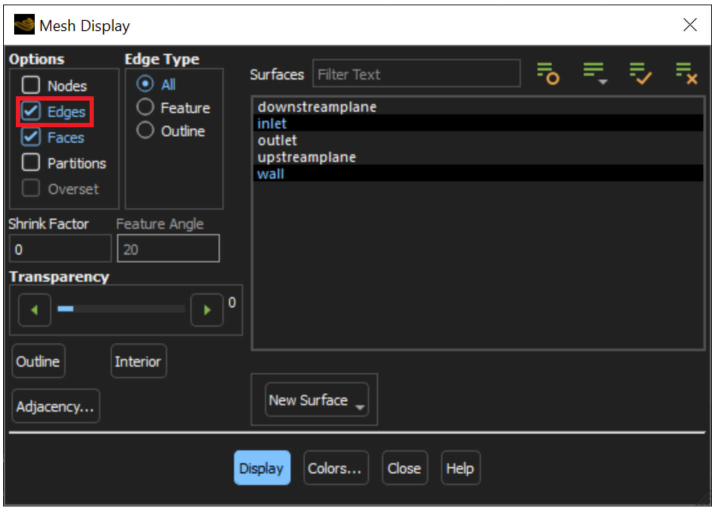


Figure 38: Set Fluent mesh display options.

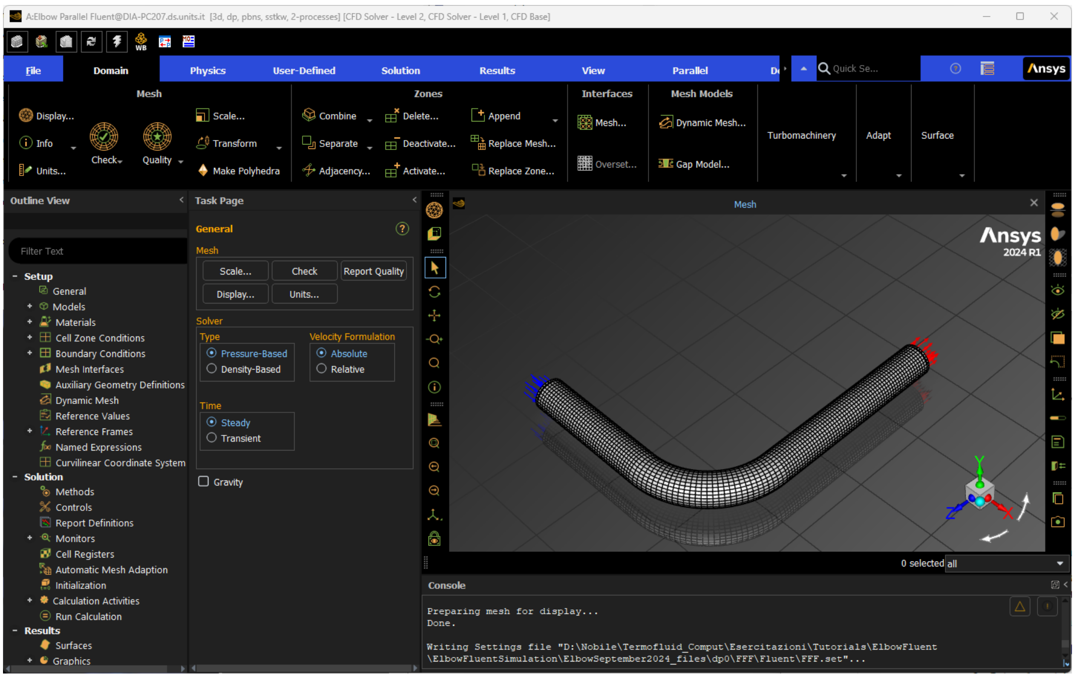


Figure 39: Fluent mesh shown in graphics window

```
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.
```

- A second check to do consists in a quick evaluation of the mesh quality. To do it, click on *Quality* $\rightarrow$ *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, 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 (

) 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 (

) 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.

  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*.

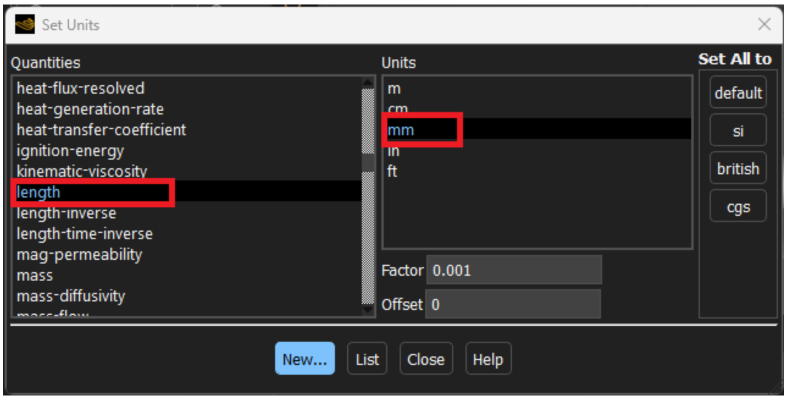


Figure 41: Set length unit in Fluent.

- 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

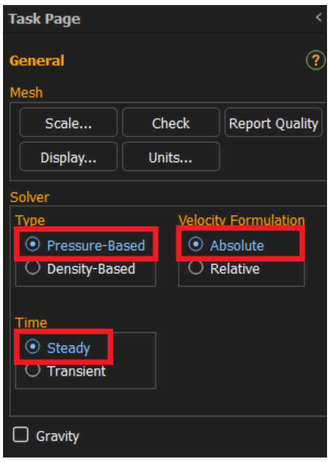


Figure 42: Solver selection in Fluent Task page.

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.
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.

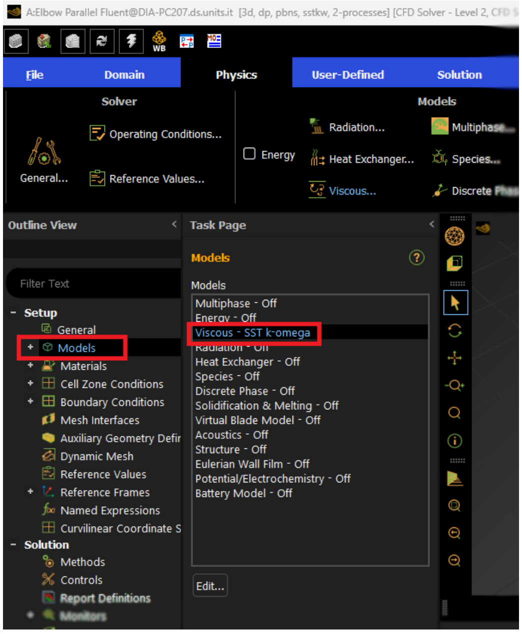


Figure 43: Checking the active models in Fluent.

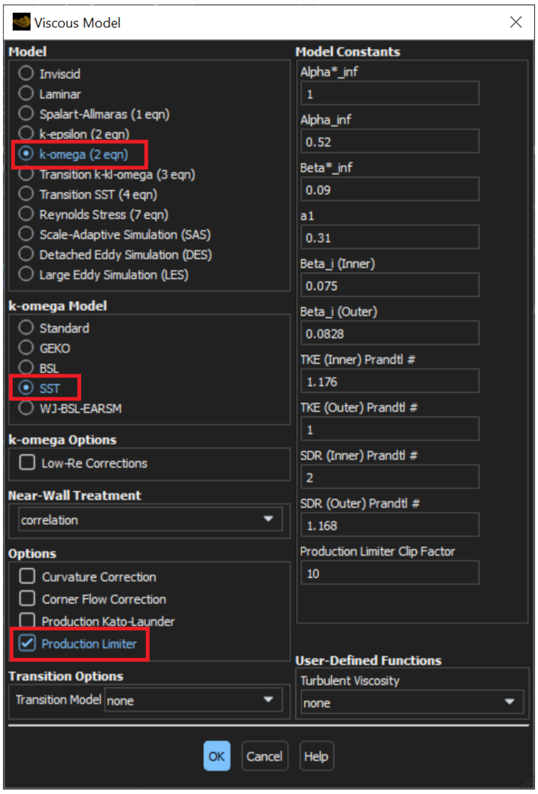


Figure 44: Set the viscous model in Fluent.

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*

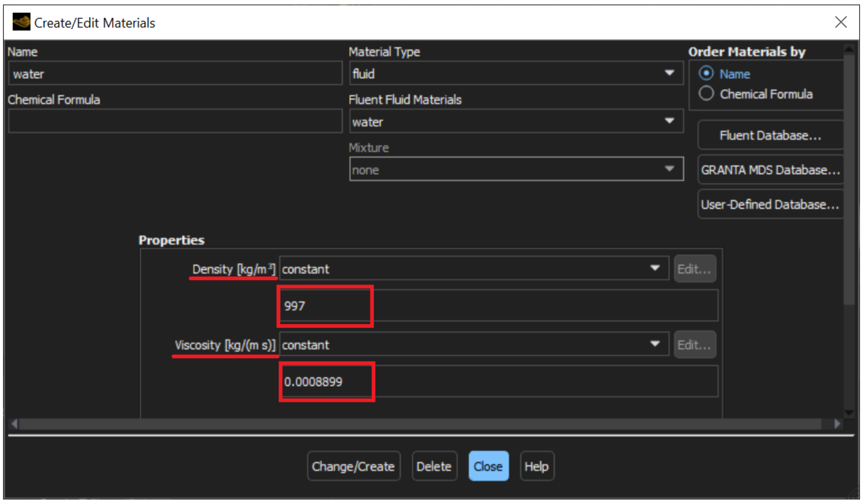


Figure 45: Inserting water properties in the Create/Edit Materials dialog box in Fluent.

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 box[3]. 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 *Boundary Conditions* task page[4]. 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.

   (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

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

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

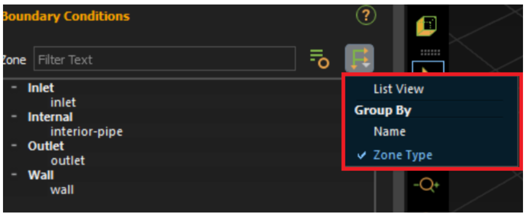


Figure 46: Boundary conditions grouped by Zone Type in Fluent.

Momentum tab as the default. Click *Apply* to close the *Velocity Inlet* dialog box.

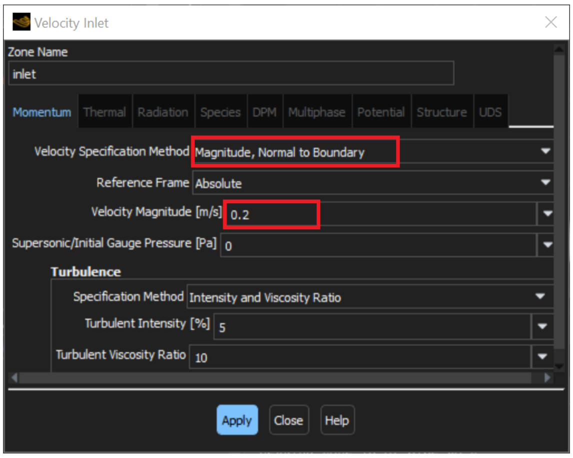


Figure 47: Inlet boundary condition as *velocity-inlet* in Fluent.

(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.

(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. Please note that the *second order upwind* scheme is selected by default for momentum and turbulent quantities.

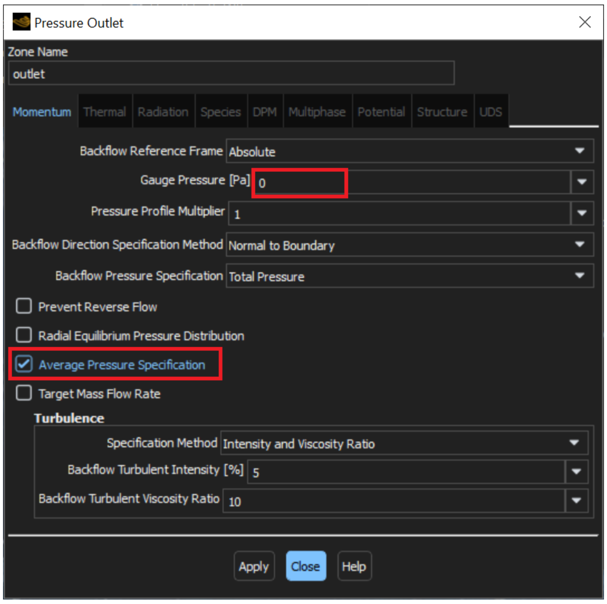


Figure 48: Outlet boundary condition as *pressure-outlet* in Fluent.

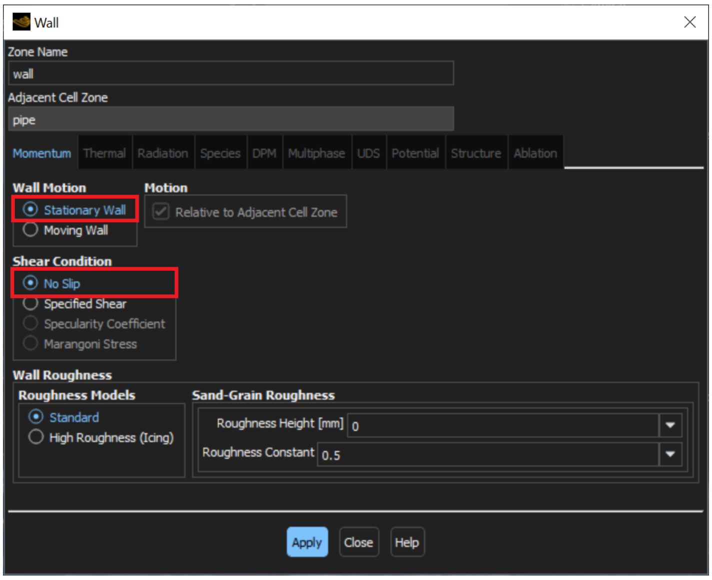


Figure 49: Wall boundary condition in Fluent.

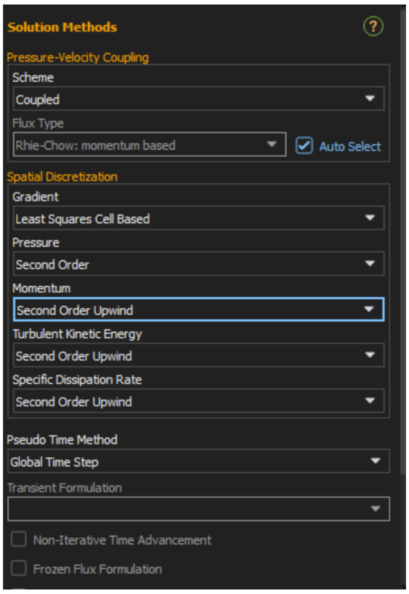


Figure 50: Solution methods selection in Fluent.

3. In the *Reports* group, click the *Residuals* icon (
). 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:

   (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 *upstreamplane*, 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.

   (b) On these two surfaces, we define the values of the - *weighted-averaged* - pressure values. For this purpose, on the main ribbon, under the *Solution* tab, select *Definitions* $\rightarrow$ *New* $\rightarrow$ *Surface Report* $\rightarrow$ *Area-Weighted Average* ..., as depicted in figure 53.

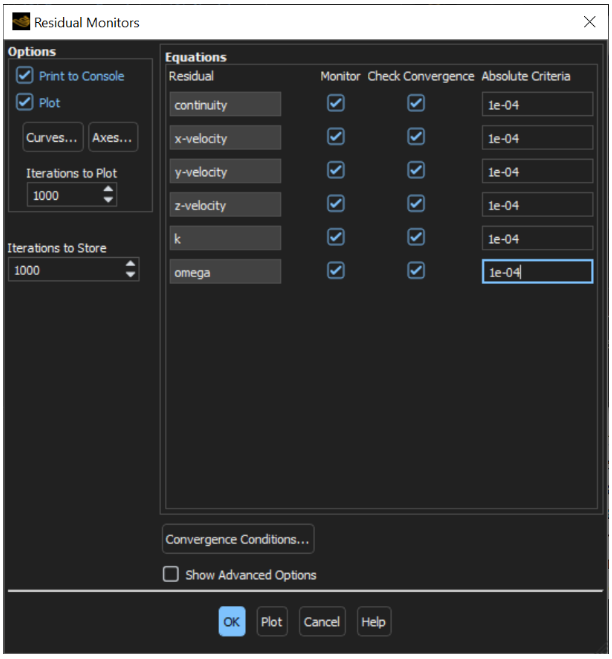


Figure 51: Convergence criteria in the *Residual Monitors* dialog box in Fluent.

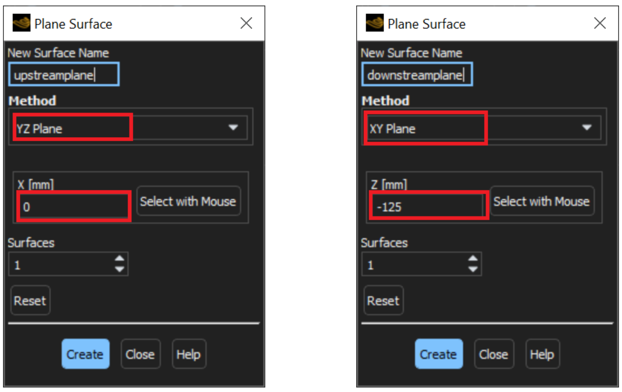


Figure 52: Upstream and downstream surfaces definition in Fluent.

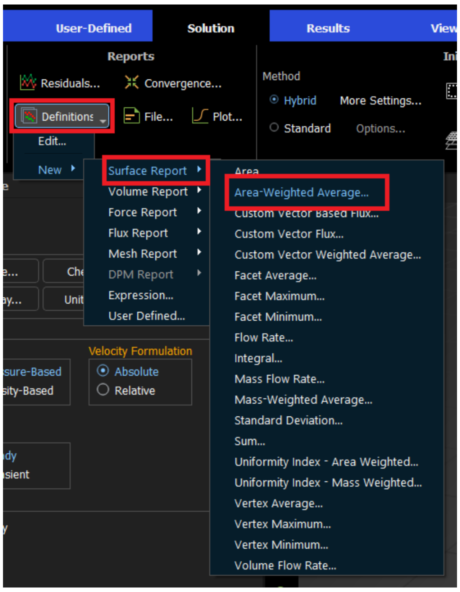


Figure 53: Definition of an Area-Weighted average Surface Report in Fluent.

On the window menu that will open, reported in figure 54, insert *pressupstream* as the name, *Pressure* and *Static Pressure* as Field Variable, checkbox the *Report File* and *Report Plot* under Create and select the *upstreamplane* under Surfaces.

Repeat the same procedure for the pressure downstream, selecting *pressdownstream* as the name, and *downstreamplane* as the surface.

(c) It is now finally w the ΔP we want to monitor during the calculation in order to establish that convergence has been achieved.
Select again *Definitions* $\rightarrow$ *New* $\rightarrow$ *Expression*. In the *Expression Report Definition* window that will open up, first set *deltap* as name and select *Functions* $\rightarrow$ *Mathematical* and then *abs*. Inside the *abs(<expr>)* insert *Report Definitions* and then "*pressureupstream - pressuredownstream*", as illustrated in figure 55. In this case, besides to checkbox *Report File* and *Report Plot*, we also checkbox *Print to Console*.

5. Before launching the calculation we have to *initialize* the solution: check that, in the *Initialization* group, the selected method is *Hybrid*, and click the *Initialize* icon(

), as shown in figure 56. After the initialization is completed - it will take around ten iterations - we are ready to start the calculation.

6. As a last step before the calculation, we can ask Fluent to check for potential problems: in the *Run Calculation...* group of the *Simulation* tab, click the *Check Case* button and, if no problems are found, the response should be like in figure 57.

7. In the *Run Calculation...* group of the *Simulation* tab set the *No. of Iterations* to, say,

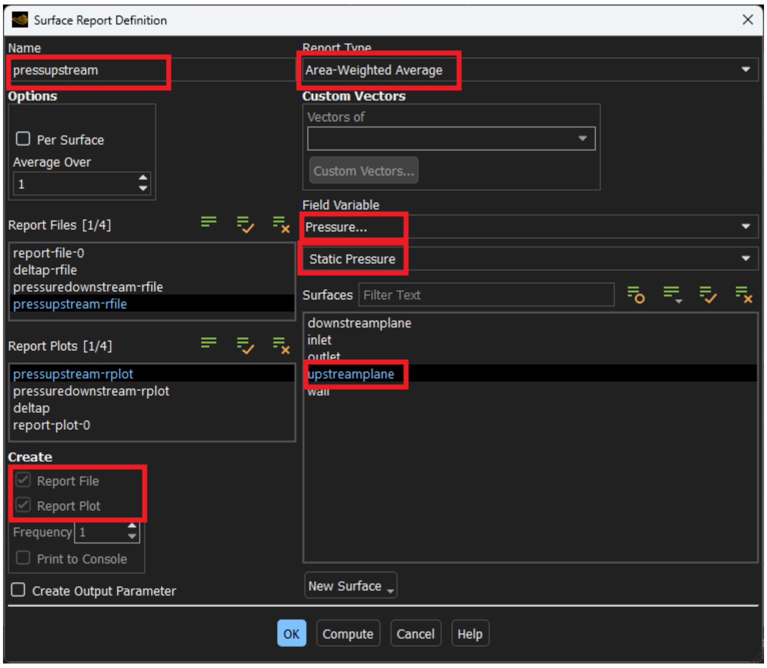


Figure 54: Definition of an upstream pressure Surface Report in Fluent.

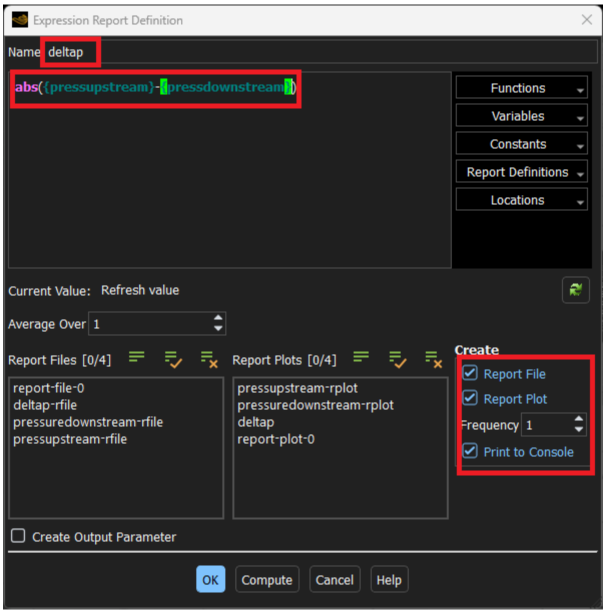


Figure 55: Definition of the pressure loss ΔP in Fluent.

Figure 56: Initialization task in Fluent.

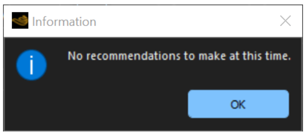
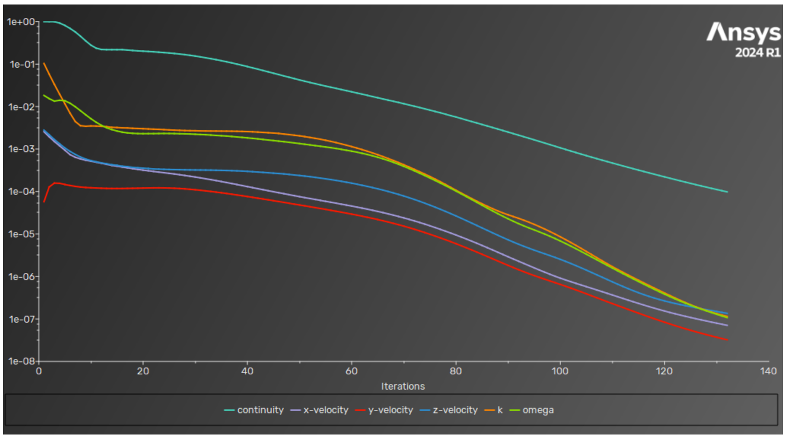


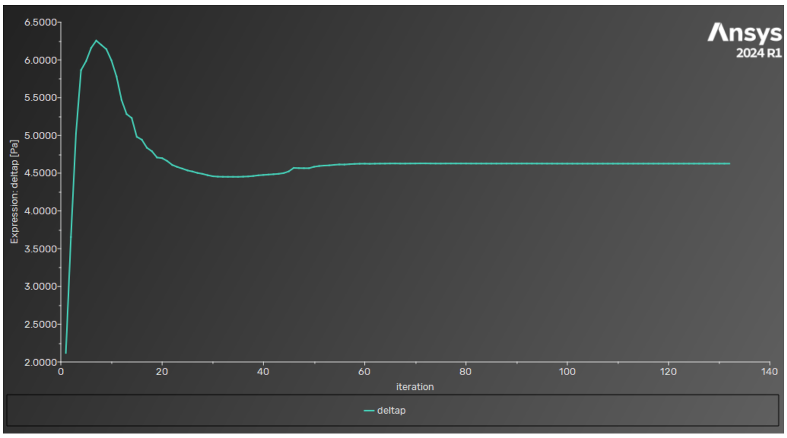
Figure 57: Fluent check case result.

500 and click the *Calculate* icon(
).[5]

The calculations will start and the graphic windows will display either the residual values or, if selected, the pressure drop ΔP. The calculation will take at most few minutes depending on the hardware available, and will complete in about 130 iterations. The reduction of the residuals and convergence of ΔP will be as in figure 58.



(a)



(b)

Figure 58: Fluent calculation completed: (a) Residuals reduction; (b) ΔP convergence.

[5]Irrespective of the maximun number of iterations set, the calculation will stop when either one of the following conditions are first met: the residual values are all below their target or the maximum number of iterations have been reached. In any case, it will be possible to *restart* the calculation from the actual state without losing any information.

3.5 Visualization and post-processing of results

It is now possible to proceed with the visualization and post-processing of the results. These activities can be done either within *Fluent* or in *Results*, the ANSYS post-processor. In what follows, we will use the former option.

1. We may first visualize the pressure field: in the *Outline View*, *Results* group, select *Graphics*, *Contours* and *New...*. In the Contours window that will open up, rename it *pressurecontour*, then select *Pressure*, *Static pressure* as variable, and *wall* as the corresponding surface, as shown in figure 59.

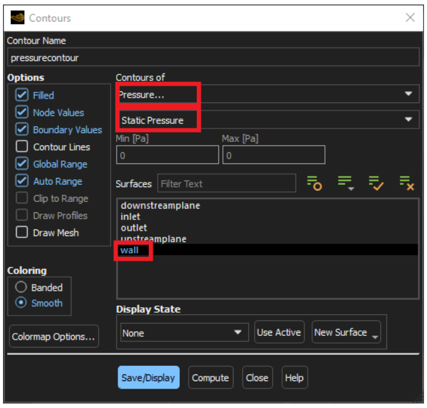


Figure 59: Fluent Pressure contour options.

Click *Save/Display* and the result will be as in figure 60.

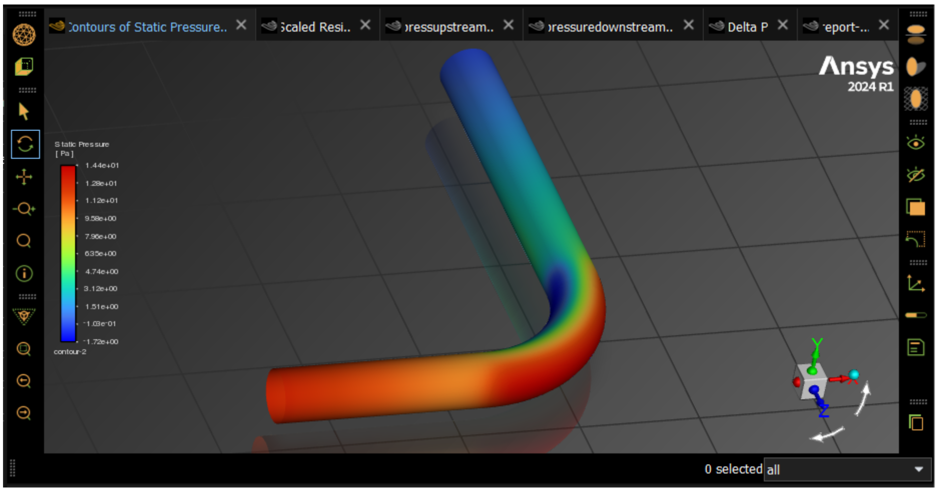


Figure 60: Pressure field on the elbow wall.

From the figure, the pressure loss within the elbow is evident, as well as the higher pressure along the external side of the pipe in comparison to the internal side.

2. In order to visualize the secondary motions, in the *Outline View*, *Results* group, select *Graphics*, *Pathlines* and *New...*. In the corresponding Pathlines window that will open up, select, as in figure 61, *inlet* in the *Release from Surfaces* list, and keep all other options as their default. Click *Save/Display* and the result will be as in figure 62.

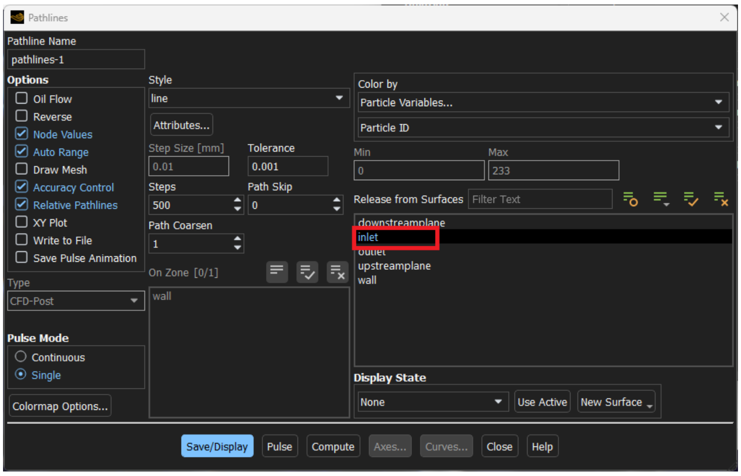


Figure 61: Fluent Pathlines options.

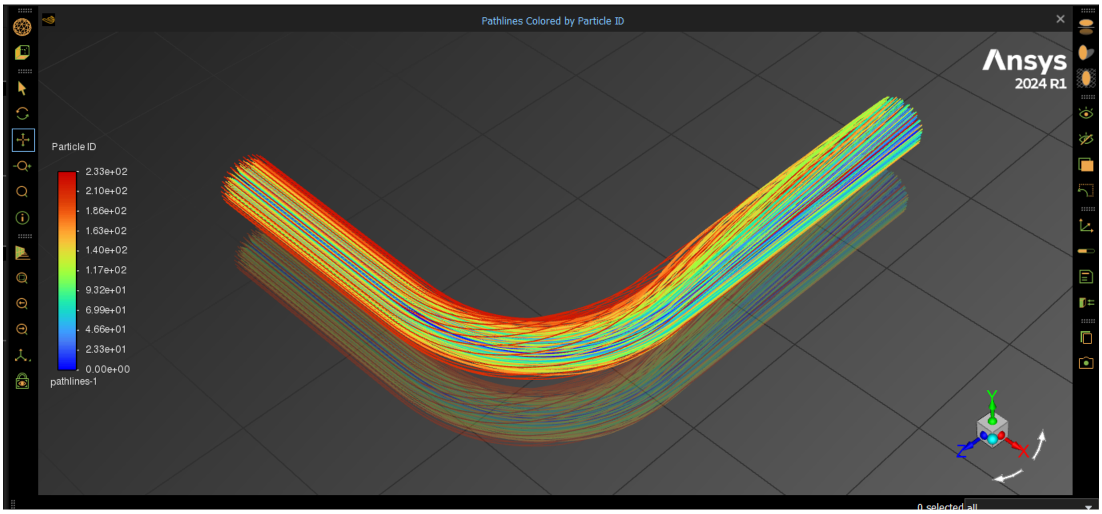


Figure 62: Visualization of secondary motions.

However, although the secondary motions are perceptible - it is sufficient, for this purpose, to observe the spiraling trend of the streamlines downstream of the curve - it is preferable to change the point of view to highlight them better. To do this, first click on the *Projection* icon (), and select *Ortographic Projection*, and then on the $-z$ axis of the triad (it is sufficient to click the z axis twice to obtain the view from $-z$). The result is illustrated in figure 63, which makes the existence of secondary motions very evident.

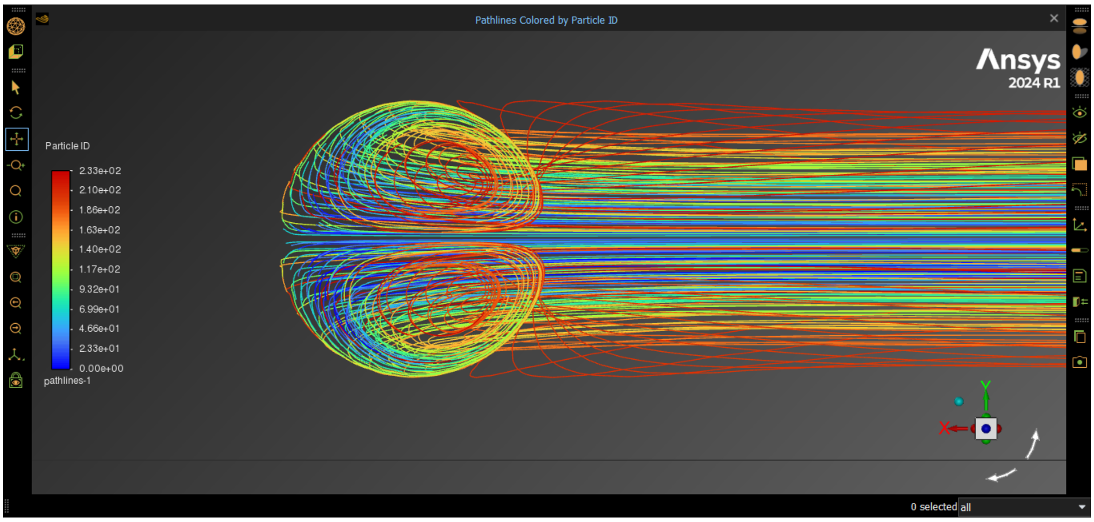


Figure 63: Clear visualization of secondary motions from $-z$.

3. Secondary motions can also be visualized by displaying velocity vectors. To do this, select again *Graphics* in the *Results* group, and then *Vectors* $\rightarrow$ *New...*. In the Vectors window that will open up, as shown in figure 65:

   - insert *vectordownstreamplane* as *Vector Name*;
   - select *downstreamplane* in the *Surfaces* list;
   - select *arrow* as *style*;
   - set 10 as *Scale* value;
   - click the *Vector Options* button: in the window that will pop up, deselect the *Z Component*, click *Apply* and then *Close*, as illustrated in figure 64.

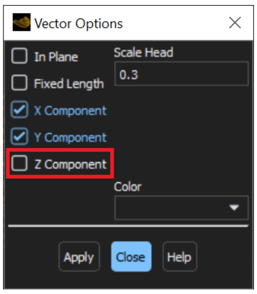


Figure 64: Fluent vector visualization options.

   Click the *Save/Display* botton and the result, after selecting the view from $-Z$, will be as depicted in figure 66.

4. The vector velocity plot clearly indicates the presence of secondary motions, however, it is not easy to compare them with the experimental findings given in figure 2. For this, it is preferable to visualize the velocity contour plot in the *downstreamplane*, in analogy

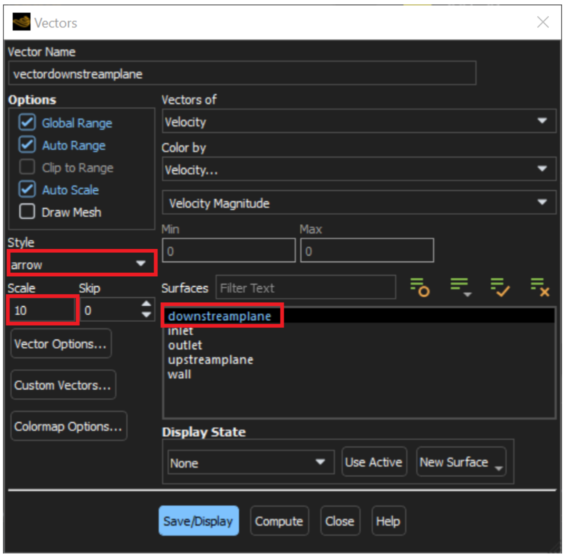


Figure 65: Selection of Fluent vector features.

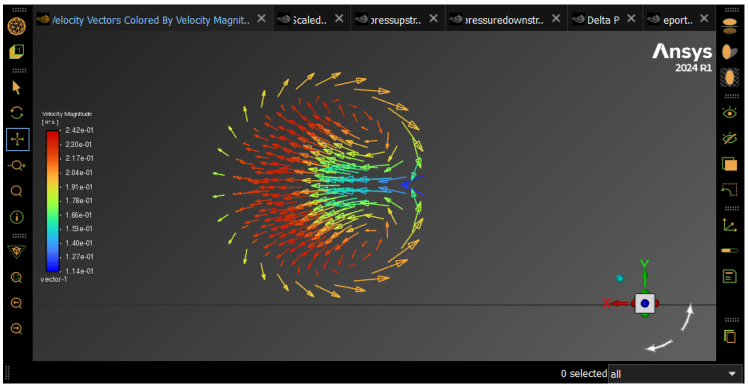


Figure 66: Visualization of secondary motions by means of vector plot.

to figure 2. To do so, in the *Outline View*, *Results* group, select *Graphics*, *Contours* and *New...*. In the corresponding Contours window that will open up, rename it as *velocity-contour* and select, as in figure 67, *Velocity...* as *Contours of* and *downstreamplane* in the *Surfaces* pane, and keep all other options as their default. Click *Save/Display* and the result will be as in figure 68 where, for convenience, also the experimental contours are provided in order to facilitate the comparison.

5. The velocity field obtained numerically certainly resembles the experimental one, however there are some differences.

6. A more quantitative analysis - that might constitutes one of the objectives of the simulation - is the evaluation of the *pressure drop* ΔP, a quantity that we have already defined and used in monitoring the convergence of the calculation. In this case it is a localized

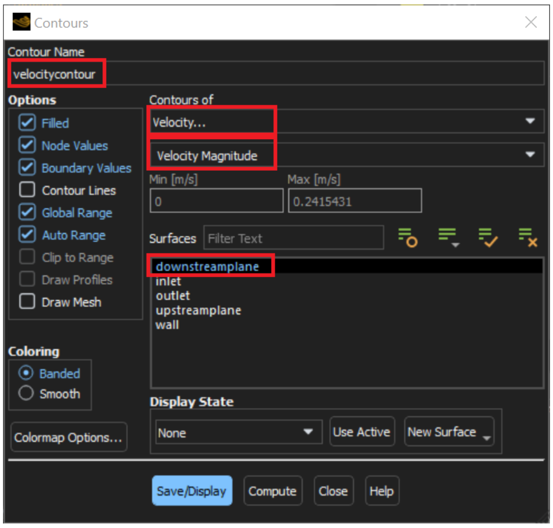
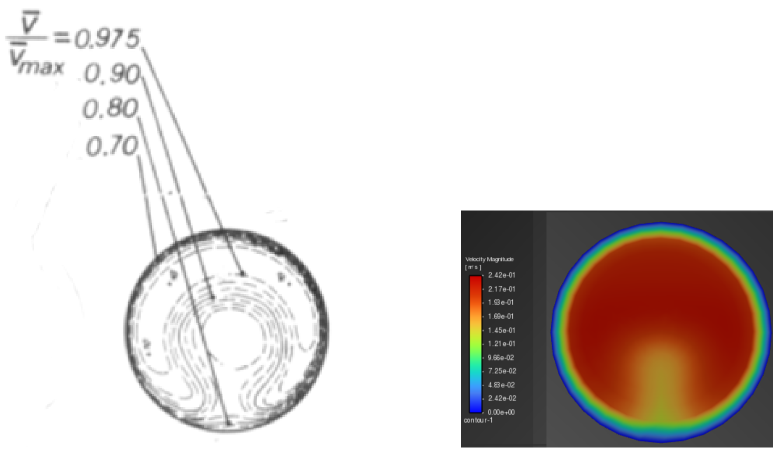


Figure 67: Fluent velocity contour options.



(a) (b)

Figure 68: Velocity contour on the *downstreamplane*: (a) experimental; (b) computed.

pressure drop, which is usually expressed with a relation like

$$\Delta p = K \frac{1}{2} \rho U^2 \tag{1}$$

where the empirical coefficient K is a function of

- r_c/D ratio, with r_c radius of curvature of the elbow axis (see figure 2);
- surface type, smooth or rough;
- Reynolds number.

The value of ΔP can be obtained by clicking, with the right mouse button in the *Outline View*, *Solution* $\rightarrow$ *Report Definitions* $\rightarrow$ *deltap* and then *Compute*, as depicted in figure 69.

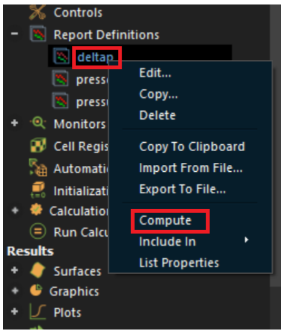


Figure 69: Compute the pressure loss ΔP in Fluent.

Doing so, it results that $\Delta P = 4.63$ [Pa]. We can now compare this value with the experimental data available in the open literature. The comparison is summarized in table 2.

Pressure drop ΔP [Pa]	
Present CFD	4.63
A. Ghetti (1977) [2]	3.50
White (2010) [3]	3.99
Ito (1960) [4]	5.76

Table 2: Comparison between the calculated pressure drop value and those reported in literature.

It can be noticed that our computed value of ΔP is in general agreement to the - somehow scattered - values from the literature, thus confirming the overall correctness of our simulation.

4 Improved accuracy

Although our results are generally in agreement with the experimental results available in the literature, they are certainly not free from errors and inaccuracies. Although we are not looking here for results free of convergence errors and grid-independent (aspects that are fully covered elsewhere), we might certainly improve the quality of our simulation by considering the following aspects (not necessarily in order of importance).

4.1 Geometry modifications

One may notice that the elbow geometry is symmetric respect the x–z plane[6]. For this reason, it is computationally convenient to perform the simulation for only *half* elbow. This can be

[6] It may be additionally noted, in figures 63 and 66, that both the streamlines and velocity vectors are also symmetric respect the x–z plane: this is reasonable and may be expected, since we have used a *steady RANS* model.

easily obtained, in *Design Modeler*, by selecting, in *DesignModeler* Menu Toolbar, *Tools* → *Symmetry*, as shown in figure 70.

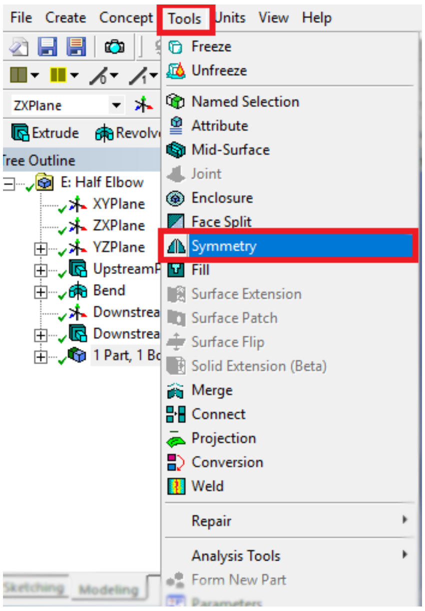


Figure 70: Selecting the *Symmetry* tool in DesignModeler.

After that, as illustrated in figure 71 in the *Symmetry Details View*, select the *ZXPlane* as a symmetry plane, and press *Generate*.

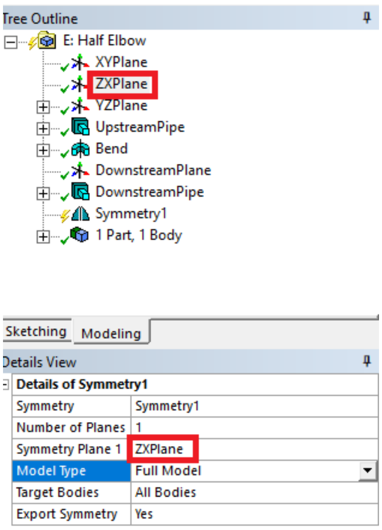


Figure 71: Choice of the Symmetry plane in DesignModeler.

The resulting modified geometry is shown in figure 72.

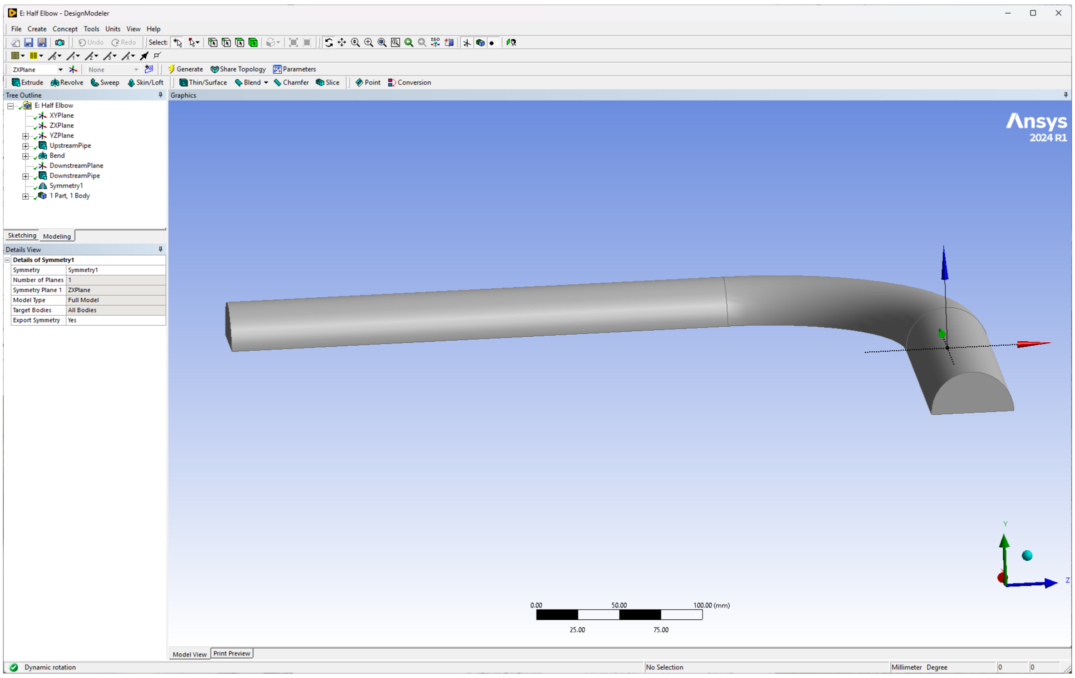


Figure 72: Modified geometry (half elbow) in DesignModeler.

4.2 Grid improvements

The grid, how we noticed before, is still inadequate, in particular for the resolution close to the wall. It can be improved, by:

1. Reducing both the *Element Size* and the *Max Size* to 2.5 mm (2.5×10^{-3} m).

2. Adding an *inflation layer*, to better capture the boundary layer close to the walls, as depicted in figure 73.

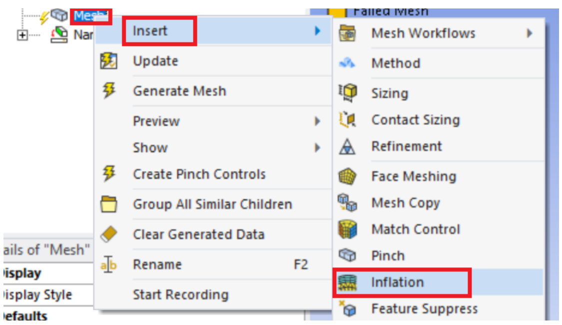


Figure 73: Adding an *Inflation Layer* in ANSYS Mesh.

In the Details menu on the left, as shown in Figure 74, verify that the *Scoping Method* is set to *Geometry Selection*, and select the face corresponding to the inlet section as the Geometry. Similarly, verify that the *Boundary Scoping Method* is set to *Geometry Selection*, and specify the edge of the face just selected as the Boundary. In this case, a reasonable choice would be to set to 12 as the number of *Maximum layers*, to select the *First layer Thickness* as *Inflation Option* and set it to 0.2 mm (0.2×10^{-3} m) and a *Growth rate* to 1.2.

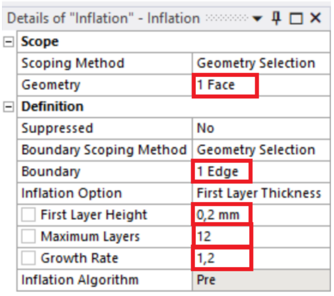


Figure 74: Details of the *Inflation Layer* in ANSYS Mesh.

3. Insert a *Method* for generating the mesh, and setting it to *Sweep Method*, as indicated in figure 75. After this, clicking on *Sweep Method* in the outline tree in the left, will open up the *Details* window on the left, where one should make the following selections, as shown in figure 76:

   (a) Select *Geometry Selection* as *Scoping Method*.

   (b) Select the *Body* of the elbow as *Geometry*.

   (c) Select *Sweep* as *Method*.

   (d) Select *Program controlled* as *Algorithm*.

   (e) Select *Manual Source and Target* as *Src/Trg Selection*. This means that the elbow will be discretized, along its axis, starting from the *Source*, in this case the inlet face, towards the *Target*, i.e. the outlet face. Select them, respectively, as *Source* and *Target* in the Details window.

   (f) Select *Number of Divisions* as *Type* and 200 as *Sweep Num Divs*.

   (g) Leave all other selections as their default value.

Once these steps are complete, visualize the mesh by clicking Generate Mesh. The final finer mesh, which has about 212.00 cells and 220.00 nodes, is illustrated in figure 77.

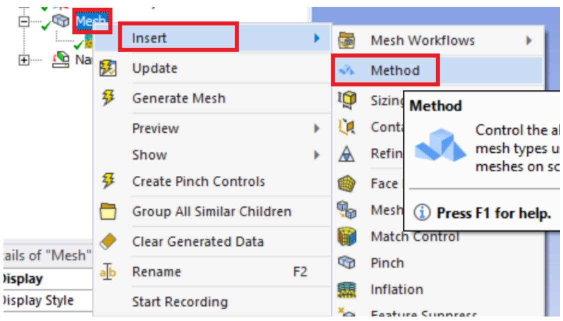


Figure 75: Inserting a mesh generation *Method* in ANSYS Mesh.

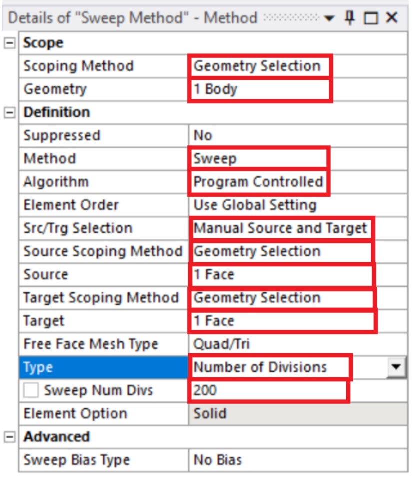


Figure 76: Inserting the generation *Method* features in ANSYS Mesh.

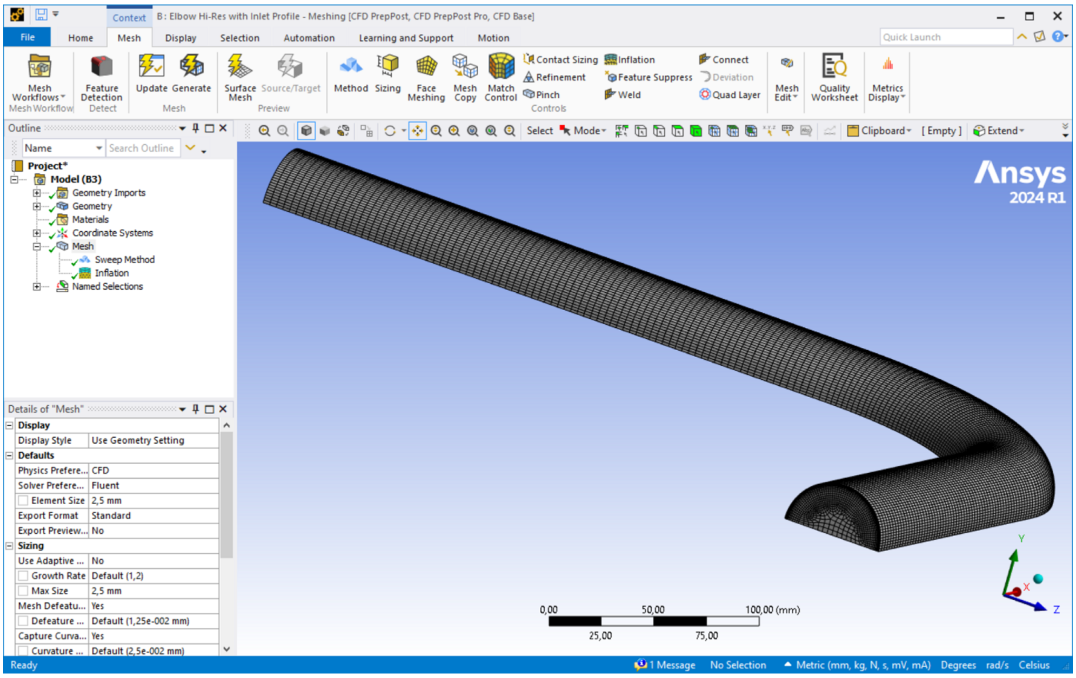


Figure 77: Final, finer mesh obtained in ANSYS Mesh.

4.3 Model and Setup modifications

1. To guarantee a better *convergence* towards the solution, it is reasonable to reduce the residuals level from 1×10^{-4} to 1×10^{-5} for the continuity, the velocity components, the turbulent kinetic energy and omega (ω), as illustrated in figure 78.

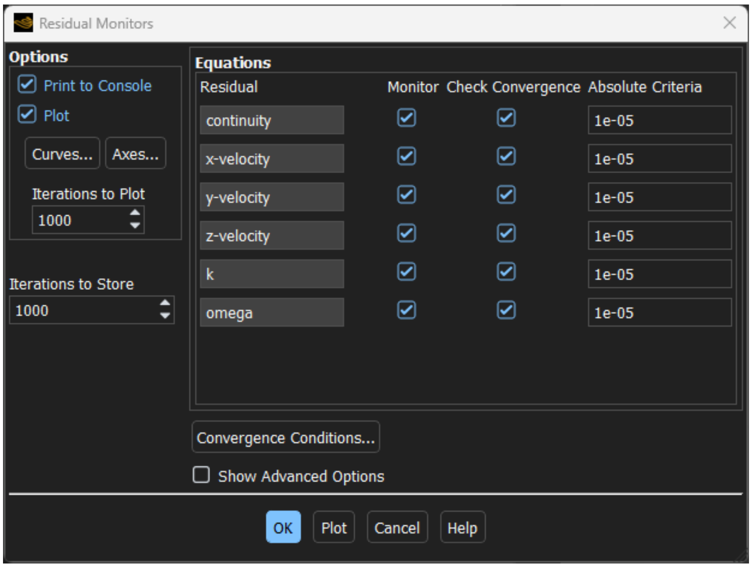


Figure 78: Reduced residual values in the *Residual Monitors* dialog box in Fluent.

2. In the previous, low-resolution calculation, we have assumed that the velocity at the inlet was *constant*, e.g. a flat profile. This is, however, an unrealistic assumption: a better choice would have been, for example, to set again a constant velocity value at the inlet and provide a longer inlet pipe, such as to allow for a complete *dynamic development* of the velocity profile before the curve [7]. This approach, unfortunately, is rather inconvenient and computationally expensive, since for a circular pipe with a value of the Reynolds number of about 10×10^3, the dynamic development length is of the order $(10 - 60)D$ [5, 6], with experimental data [7] indicating a value $\approx 30\,D$.

3. A better, more realistic approach, would consider to assign a *fully developed velocity profile* at the inlet. There are several formulas for the turbulent, fully developed velocity profile for smooth circular tubes, see e.g. [6] for the most common, but in our case, since we provide an inlet pipe where the velocity profile may somehow *adjust* to this problem, it is sufficient to consider one of the simpler ones, due to Prandtl [8]:

$$\frac{U}{U_{max}} = \left(\frac{y}{D/2}\right)^{\frac{1}{n}} \tag{2}$$

$$\frac{U_m}{U_{max}} = \frac{2\,n^2}{(n+1)\,(2n+1)} \tag{3}$$

[7]The flow is said to be *dynamically fully developed* if the velocity profile no longer changes with increasing x, e.g. the distance from the inlet section. The distance from the entrance at which this condition is achieved is termed the *hydrodynamic entry length*, $x_{fd,h}$.

where, in eq. (2), U_{max} is the maximum value of the velocity at the centreline, U is the local value of the velocity and y is the radial distance measured from the duct wall.
The exponent n varies slightly with Re (see [6] for further details), and in our case it can be assumed $n \approx 6.2$.
Therefore, from (3), it results U_{max} = 0.251 [m/s], and eq. (2) can be inserted directly - or using the expression editor - as inlet boundary condition in Fluent, as shown in figure 79.

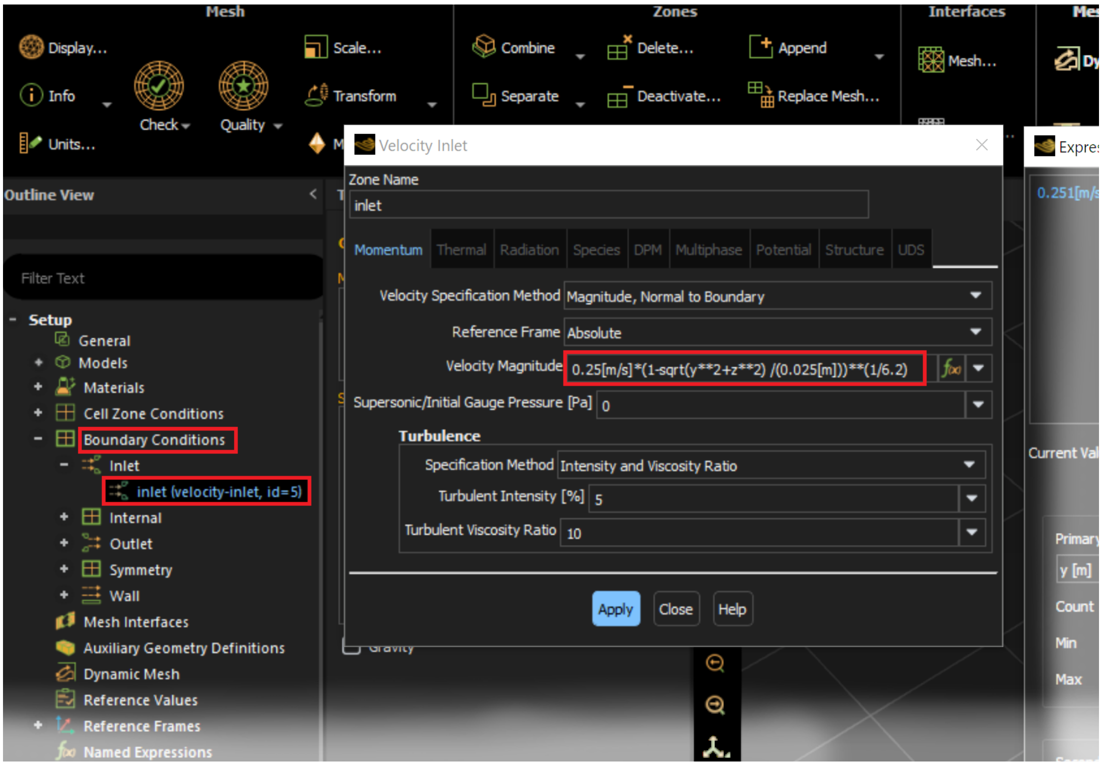


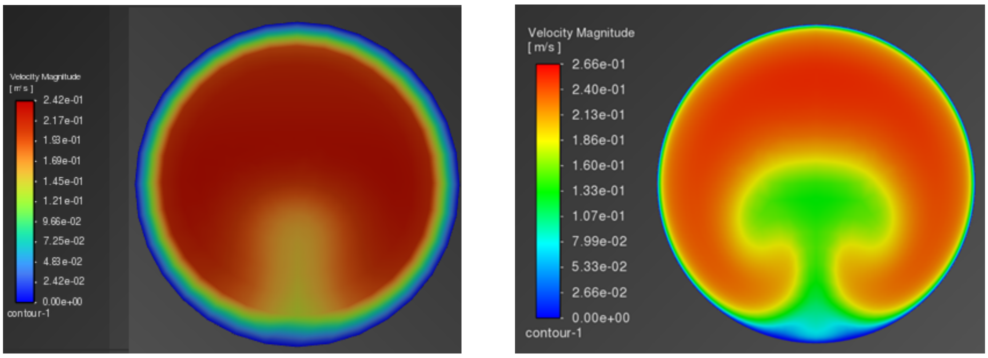
Figure 79: Setting a velocity profile as inlet boundary condition in Fluent.

Once all these improvements and modifications are done, the calculation can be repeated and the result is $\Delta P = 4.67$ [Pa], a value that differs by less than 1 % from the result obtained previously with a coarser grid and different inlet boundary condition.

However, by comparing the velocity contour on the *downstreamplane* obtained now with that obtained previously on the coarser grid, as depicted in figure 80, it is clear that the higher resolution provided by the new mesh allows a more realistic prediction of the velocity field (compare these plots with the experimental contour of figure 2).

5 Concluding Remarks and Further Activities

As has been demonstrated, the overall accuracy of numerical results depends on the numerical simulation settings (the convergence criterion), on the modeling choices (a fully developed velocity profile at the inlet rather than an unrealistic flat velocity profile), and on the quality



(a) (b)

Figure 80: Velocity contour on the *downstreamplane*: (a) coarser grid; (b) refined grid.

and resolution of the computational grid. There is also the issue of *modeling error*, not addressed in this simple tutorial. Among the possible activities left to the interested student, we highlight:

- Use of other turbulence models, such as, for example, the Realizable $k - \varepsilon$ model or a Reynolds Stress Model (RSM).
- Evaluation of the *Convergence error*.
- Analysis of *Grid independence*, i.e. identify the minimum level of grid resolution which guarantees a *grid-independent* solution.
- Repeat the calculation on a *block-structured* computational grid.

References

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

[2] A. Ghetti, IDRAULICA, *Cortina Ed.*, 1977.

[3] F. M. White, FLUID MECHANICS, *McGraw-Hill*, 7th Ed., 2010.

[4] H. Ito, Pressure losses in smooth pipe bends, *ASME J. Basic Engineering*, 131-140, 1960.

[5] T. L. Bergman, A. S. Lavine, F. P. Incropera, D. P. Dewitt, FUNDAMENTALS OF HEAT AND MASS TRANSFER, 7th Ed., Wiley, 2011.

[6] S. Kakaç, R. K. Shah and W. Aung, *HANDBOOK OF SINGLE-PHASE CONVECTIVE HEAT TRANSFER*, Wiley, 1987.

[7] A. R. Barbin and B. Jones, Turbulent Flow in the Inlet Region of a Smooth Pipe, *J. Basic Eng.*, **85**, pp. 29-34, 1963.

[8] L. Prandtl, Über den Reibungswiderstand strömender Luft, *Ergbnisse der Aerodynamics Versuchstalt zu Göttingen*, Edition 3, pp. 1-5, 1927.