
Hands-on session 3 – Mesh Generation and Application to Fully Developed Laminar Pipe Flow


Abstract

This session will introduce you to the creation of a simple geometry with SpaceClaim® (already installed with Fluent) and generate a computational grid with Fluent® using the meshing option. For this exercise we will study the classical **fully developed laminar pipe** case at Reynolds number equal to 500 with air at density 1.225 kg/m^3 and dynamic viscosity $1.789 \times 10^{-5} \text{ [kg/m-s]}$. These are the default values in Fluent. The diameter of the pipe is 50 mm. In addition, we will confirm the difference between two different type of mesh, that is to say the Hexahedral (named Hexcore in Fluent) and the Tetrahedral grids.

Goal

The aim of this hands-on session is to learn how to generate a simple geometry and mesh it including boundary layers. In addition, the student will compare two different approaches to meshing (Hex and Tet) and become aware of their quality and the influence for the number of iterations it takes to reach convergence. The student should refer to “Hands-on Session 22” for some of the functionality that will not be repeated in the current document. The following aspects will be explained in detail:

1. Create a simple geometry – 2D-long pipe
2. Generate a computational grid (Hex and Tet)
3. Simulate a fully developed pipe flow
4. Compare the results with the analytical profile
5. Compare skewness, orthogonality and number of iterations to reach convergence for each mesh.


Author	CFDLab@Energy	 Page 1 of 24
CFD for Nuclear Engineering	Session 2	

Hands-on session 3 – Mesh Generation and Application to Fully Developed		
	Laminar Pipe Flow	1
1	Introduction	3
2	Geometry Generation in ANSYS SpaceClaim.....	4
2.1	Start SpaceClaim®	4
2.2	Draw a Circular Sketch.....	4
2.3	Extrude to Generate a Pipe.....	5
2.4	Assign Names to the Boundary Surfaces.....	6
3	Mesh Generation in Fluent Meshing.....	8
3.1	Watertight Workflow	8
3.1.1	Import Geometry.....	9
3.1.2	Add Local Sizing	9
3.1.3	Generate the Surface Mesh	9
3.1.4	Describe Geometry	10
3.1.5	Update Boundaries.....	11
3.1.6	Update Regions	11
3.1.7	Add Boundary Layers.....	11
3.1.8	Generate the Volume Mesh.....	12
3.2	Visualization and Modification of the Settings.....	12
4	Set-up the Case in Fluent.....	14
4.1	Analyze the Problem.....	14
4.2	Check the Mesh and Set-up Models.....	14
4.3	Define Periodic Boundary Conditions	14
4.4	Define the Solutions	16
4.5	Define Reports.....	17
4.6	Run the Simulation	19
4.7	Results.....	20
4.7.1	Visualization of the Velocity Contours	20
4.7.2	Compare the Velocity Profile	21
4.7.3	Export the Velocity Profile for Comparison.....	23
5	Compare two Different Grids	24

1 Introduction

The fully developed laminar pipe flow will be used to test the types of mesh that we can create in ANSYS Fluent. The fully developed laminar pipe is the fundamental and simplest case to build-up the knowledge with turbulent cases and with more complex geometries. It will be also a good way to strengthen the knowledge of the code itself and expand to SpaceClaim modeling as well as Fluent Meshing.

Note: It is advised to prepare an excel file to summarize the fluid properties and boundary conditions for the resolved variables (pressure, x- and y-velocity, temperature). Some of these data may require calculation depending on the information given in the case description.

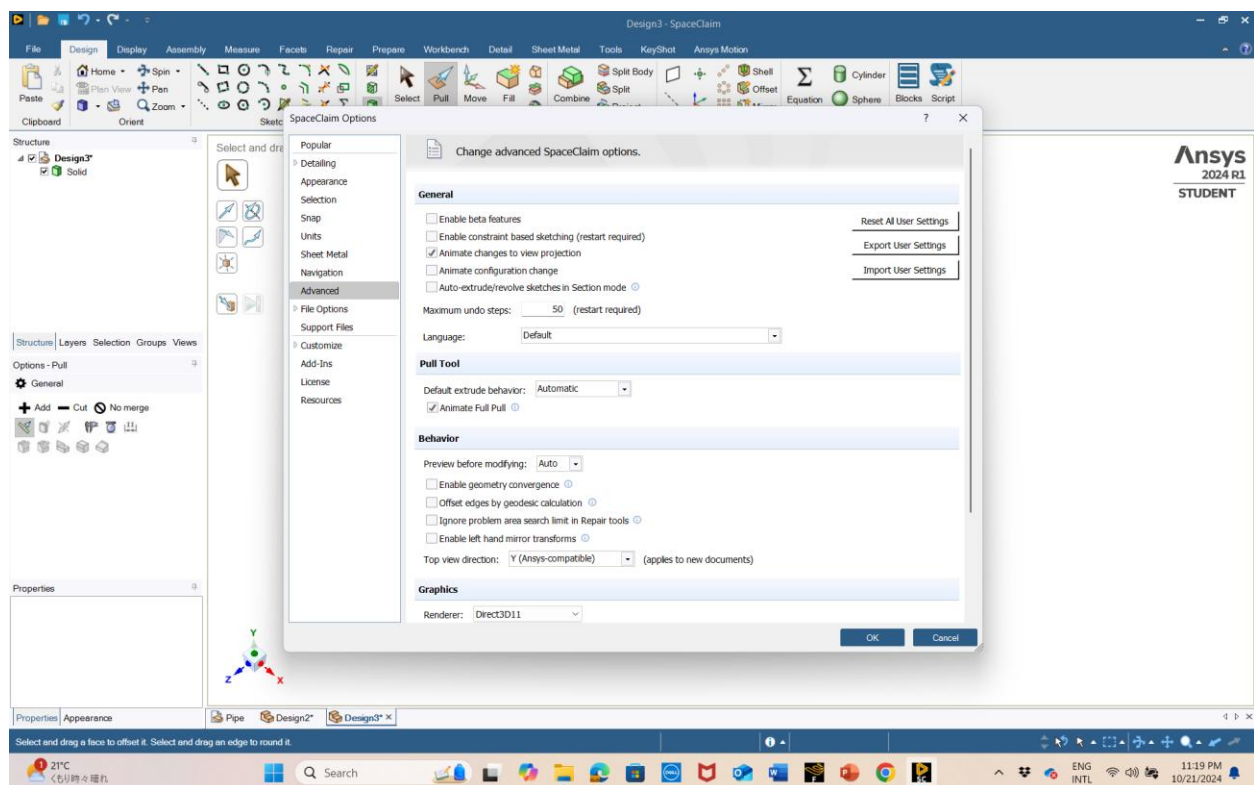
Author	CFDLab@Energy		Page 3 of 24
CFD for Nuclear Engineering	Session 2		

2 Geometry Generation in ANSYS SpaceClaim

2.1 Start SpaceClaim®

To start SpaceClaim type SpaceClaim in your Search Bar and Click on the App. SpaceClaim works as a typical CAD generation tool in which you generate a 2D sketch which can be extruded and later, optionally, cut. Combining sketches, extrusions and cut it is possible to create any kind of geometry.


SpaceClaim does not include as default the possibility to modify the geometry by clicking on a feature, hence before starting it is required to click on File -> SpaceClaim Options

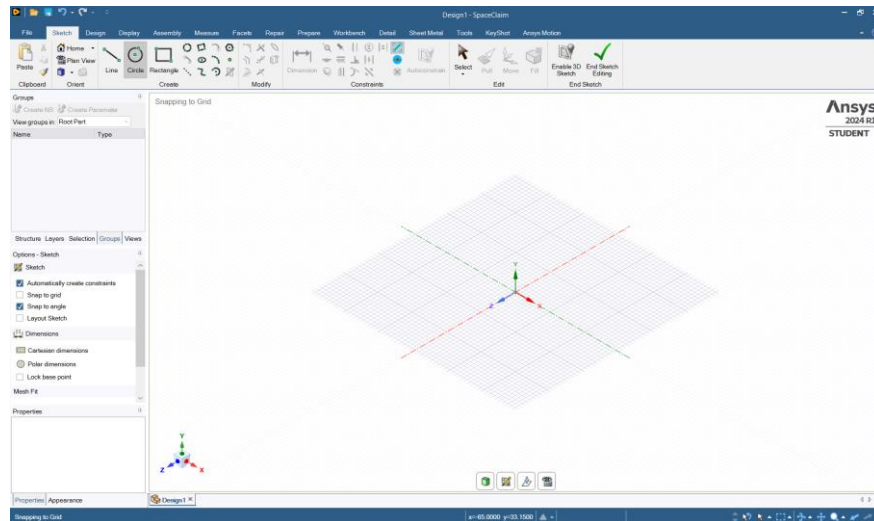



In the Options window, click Advanced and disable in General, “Enable constraint based sketching (restart required)”. As it is mentioned, once you disable and click OK, the code will request to restart. Hence, restart.

2.2 Draw a Circular Sketch

SpaceClaim will open on the empty view showing the grid and the axes in an isometric position.

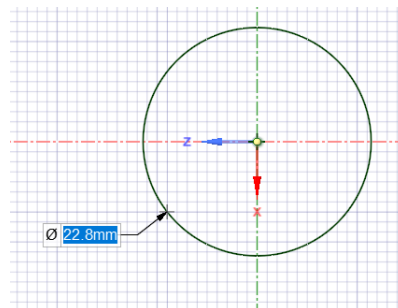
Author	CFDLab@Energy	 Page 4 of 24
CFD for Nuclear Engineering	Session 2	



To draw a circle it is better to move the orientation orthogonally to the grid by clicking the Plan View Icon () below the grid or by pressing the letter “v”.

Select “Circle” in the tool bar and click in the center of the axis.


As you move the mouse pointer far away from the center you will notice that a circle will be displayed and its current diameter will be identified in mm.



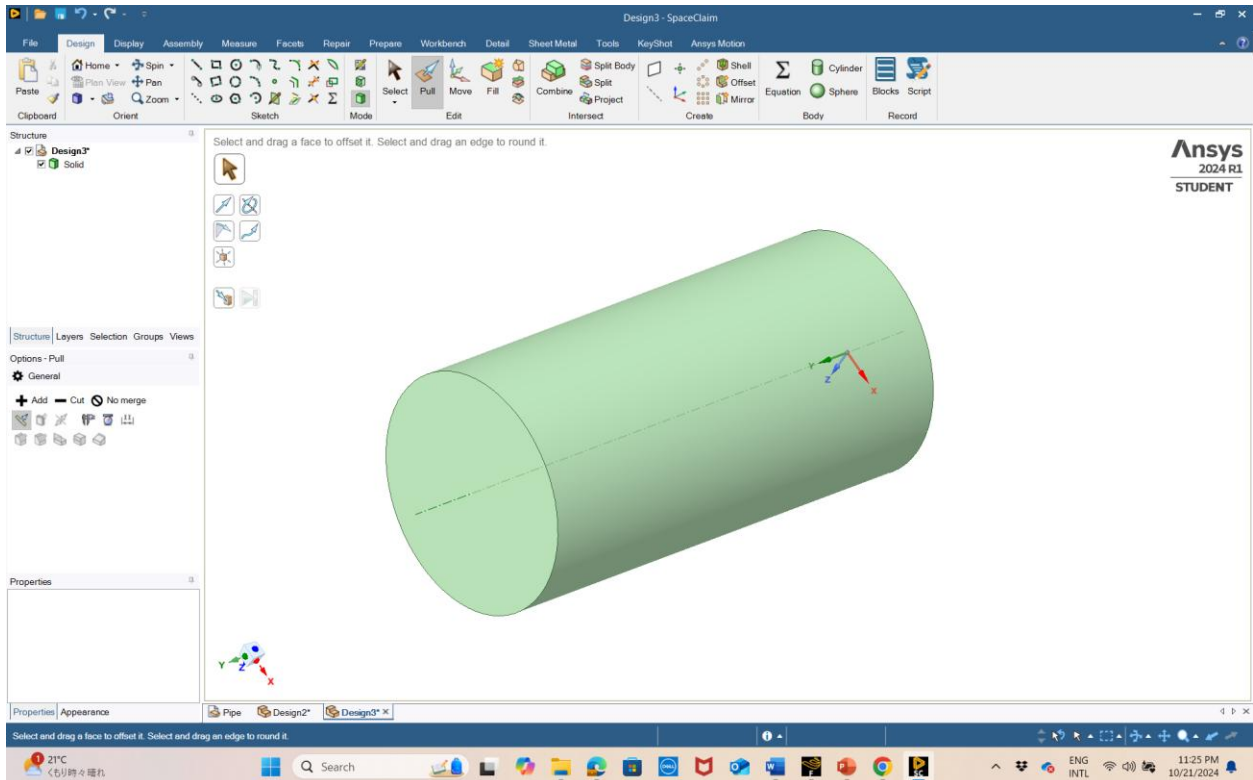
If you want to define the diameter of the circle press the “spacebar”. Now the mouse is “disconnected” from the circle and the user can input the value of the diameter. Please input 50.0 and press enter.

Next, “End Sketch Editing” on the toolbar top-right position.

2.3 Extrude to Generate a Pipe

Author	CFDLab@Energy	 Page 5 of 24
CFD for Nuclear Engineering	Session 2	


The code will automatically change to “Pull” mode. If you move with your mouse on the surface you will be automatically able to change the extrusion of your object. Click on the surface and extract, using the spacebar, for 100m. The pipe will be created 2D long.

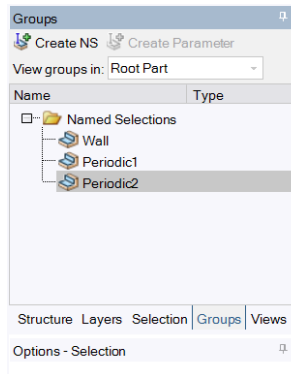


The pipe generation is completed.

2.4 Assign Names to the Boundary Surfaces

Select “Groups” on the left tab and select the pipe wall. Click in the Groups area “Create NS” meaning “Create a New Selection”.

Author	CFDLab@Energy	 Page 6 of 24
CFD for Nuclear Engineering	Session 2	

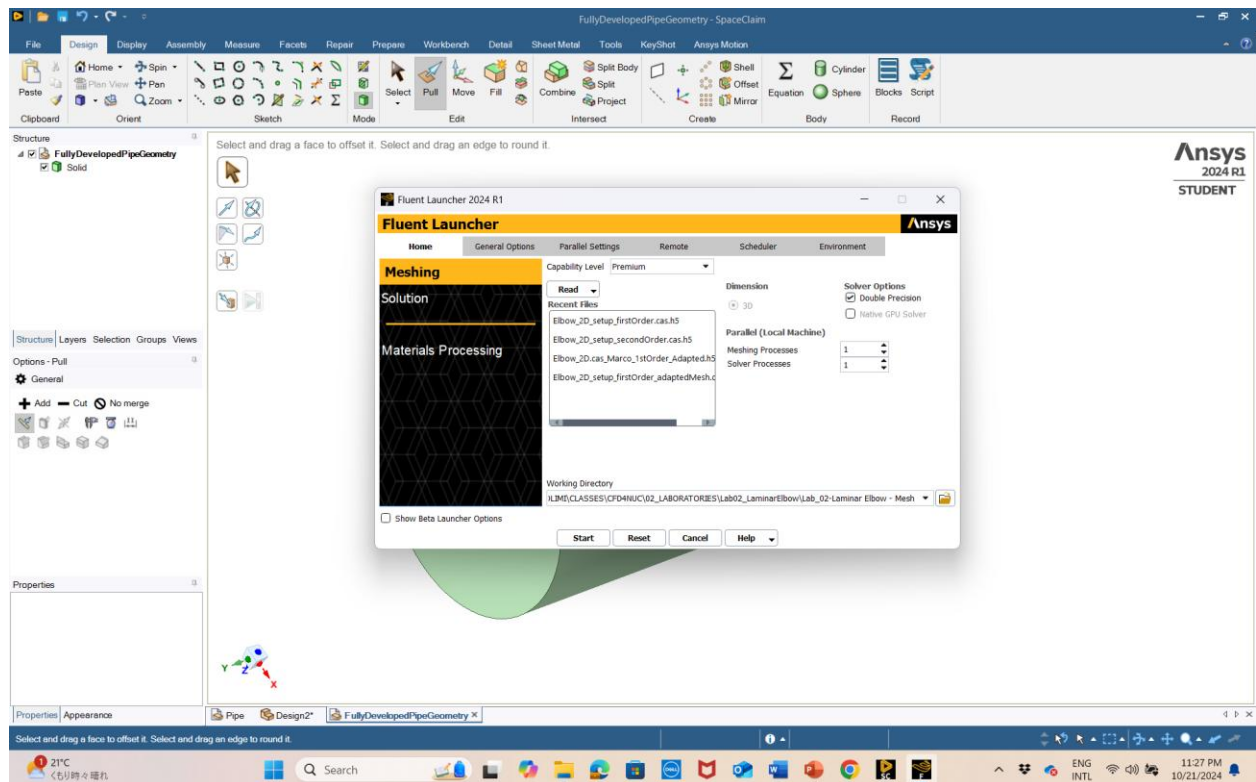


Modify the name to Wall and repeat for the other two faces of the pipe naming them respectively “Periodic1” and “Periodic2”. Periodic is the terminology used when we want to apply two faces to equal conditions for velocity, pressure, etc... this is often the case for fully developed flows (e.g. pipes, channels) or periodic flows (e.g. parallel jets).

Save As -> FullyDevelopedPipeGeometry.scdoc (or any name of your choice)

SpaceClaim Student version can create only scdoc files. However, this is enough for what we need.

3 Mesh Generation in Fluent Meshing




Meshing in Fluent is only 3D so no need to worry about the selection.

3.1 Watertight Workflow

Select “Watertight” geometry. Watertight workflow can be used in case of clean CAD with no leaks or other geometry issues, which is the present case given the very simple geometry.

Watertight geometry requirements:

- 3D geometry
- Single body or multiple bodies with shared topology applied either at mesh level or at CAD level

Author	CFDLab@Energy	 Page 8 of 24
CFD for Nuclear Engineering	Session 2	

3.1.1 Import Geometry

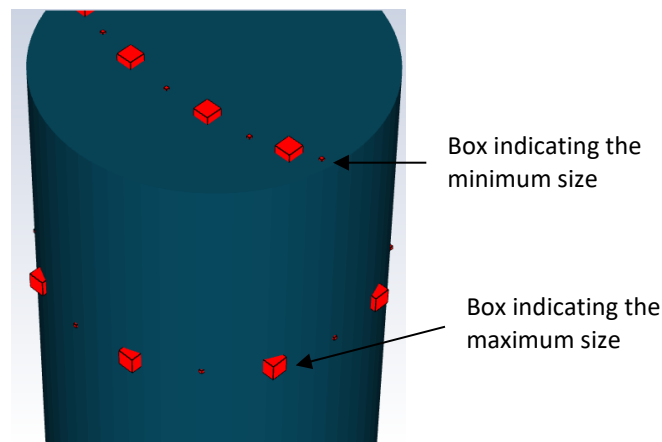
Click on the Import Geometry on the workflow, leave all the default options and navigate to the location of the .scdoc and click import. Every time a step is completed successfully a green check will appear on the left side of the step and the software will move automatically to the next one.

3.1.2 Add Local Sizing

Leave “No” and Update. Local sizing option will be presented in future sessions.


3.1.3 Generate the Surface Mesh

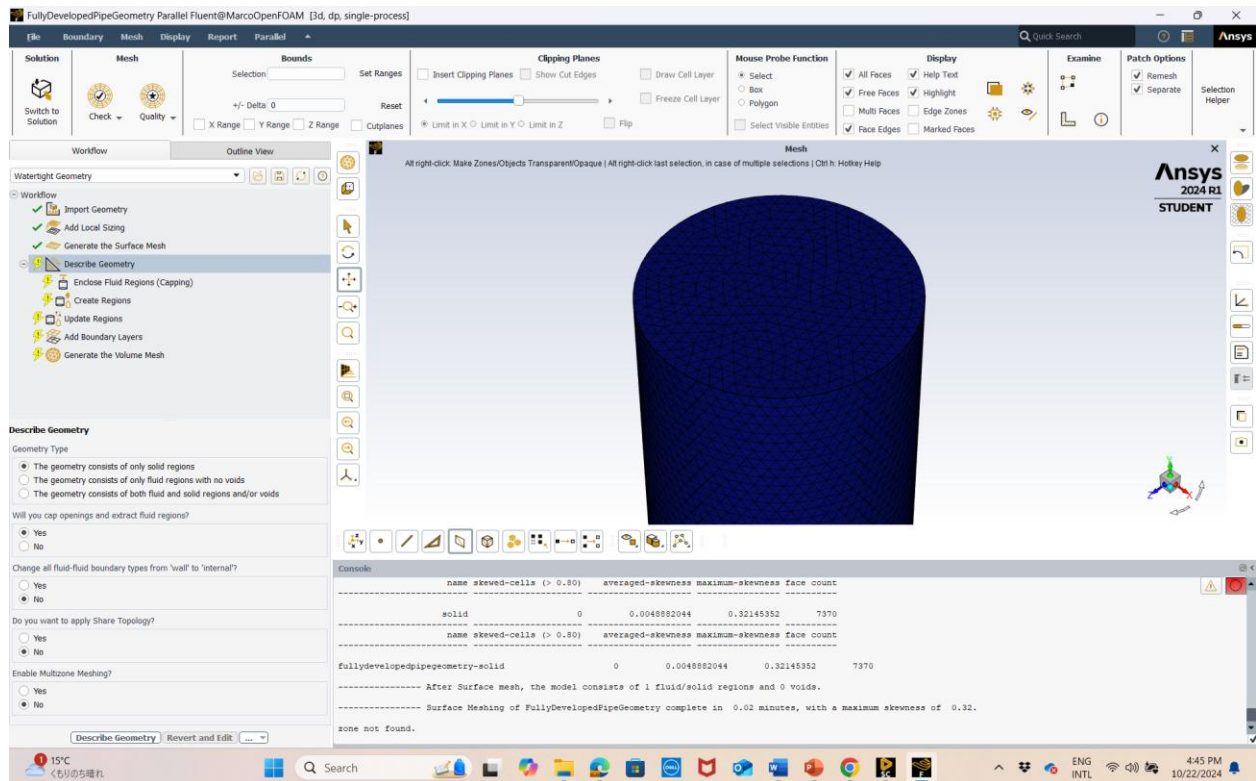
At this point we can select the Maximum and the Minimum size of the surface cells. Fluent can generate only triangular surface cells. Some red boxes will be drawn on your domain to visually indicate the maximum size and minimum size of the surface cells. Careful! The minimum size might be too small to be seen with the naked eye!



They need to be changed depending on the problem to be solved and the computational capability available. Change the minimum size to 1.0 mm and leave the maximum size to 2.5 mm and click Generate the Surface Mesh.

The surface mesh will be created with Triangles and displayed on the many window. The average and maximum skewness of the surface cells will be reported in the Console area.

Author	CFDLab@Energy		Page 9 of 24
CFD for Nuclear Engineering	Session 2		



3.1.4 Describe Geometry

Geometry Type

This is a very simple geometry so select “The geometry consists of only fluid regions with no voids”.

Change all fluid-fluid boundary types from ‘wall’ to ‘internal’?

As there is only a single fluid this selection will have no influence. Please leave the default value.


Do you want to apply Share Topology?

This is an option to force a conformal mesh on two interfacing surface belonging to multiple parts. As we have a single part please select “No”.

Enable Multizone Meshing?

Select “No”.

Click “Describe Geometry”

Author	CFDLab@Energy		Page 10 of 24
CFD for Nuclear Engineering	Session 2		

3.1.5 Update Boundaries

In this section the three boundaries created in SpaceClaim will appear and automatically assigned as walls. As we will see later boundary layers are usually generated on the walls, so it is important to modify the Periodic1 and Periodic2 boundaries as not-walls. Technically they should be assigned to “Periodic” in the Boundary Type, however this option is missing in the meshing mode of Fluent. It will be present in the solution model so we need to remember to update it.

However, as long as we modify from Wall to another boundary, the boundary layer will not be generated. Change tentatively to Inlet for one face and Outlet for the other face. Leave Walls as Wall.

3.1.6 Update Regions

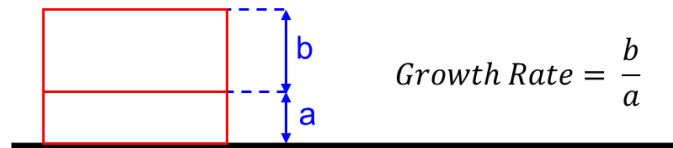
Change the name of the region to “Fluid” and select region type as “fluid”.

Click Update Regions.

3.1.7 Add Boundary Layers


The generation of a correct boundary layer is fundamental to obtaining a correct simulation in particular in a turbulent flow. For a laminar flow it is necessary to have enough refinement to compute the velocity gradients which are the highest the wall.

For this example, select “Offset Method Type” as “Uniform”, number of layers “5”. Leave the “Growth Rate” as 1.2 and change the first height 2 mm. The meaning of **Growth Rate** is the ratio between the next boundary layer cell to the current boundary layer cell, as in the figure below:



Click “Add Boundary Layer”. The workflow will not shift directly to “Generate the Volume Mesh” as the user might want to add another boundary layer setting, which is not our case for the moment. Hence, select “Generate the Volume Mesh”.

Based on the definition of Growth rate and the size of the first cell close to the wall (indicated with the letter a), would you be able to evaluate the total length of the boundary layer?

Author	CFDLab@Energy		Page 11 of 24
CFD for Nuclear Engineering	Session 2		

3.1.8 Generate the Volume Mesh

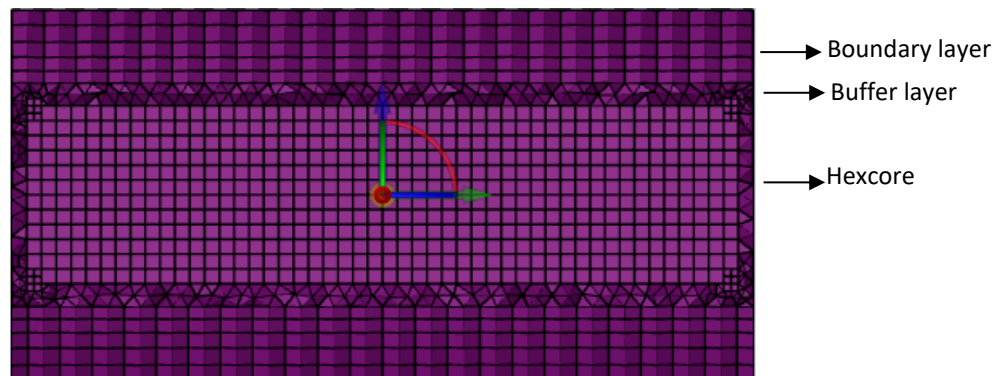
Solver will be Fluent. Fill with, select Hexcore. This will generate Hexahedral mesh in the volume. The boundary layer is hexahedral by definition so we might expect a non-conformal mesh to be generated in the volume. However, Fluent does not generate non-conformal mesh in this step (it can generate during a refinement) and hence, to keep it conformal it will generate a transition layer to cover the rapid transition between the finer boundary layer with the coarser bulk region. This transition layer is named below “Buffer Layer”. By selecting “Hexcore” for the volume cells the buffer layer will be created with Tetrahedral. If the volume mesh was selected as “Poly-hexcore” the transition (or buffer layer) would be created as polyhedral cells.

Leave buffer layers as 2, Minimum Cell Length 1 mm and Maximum Cell Length 2 mm. This size represents the maximum and minimum cell size in the bulk region.

Finally click, “Generate the Volume Mesh”.


3.2 Visualization and Modification of the Settings

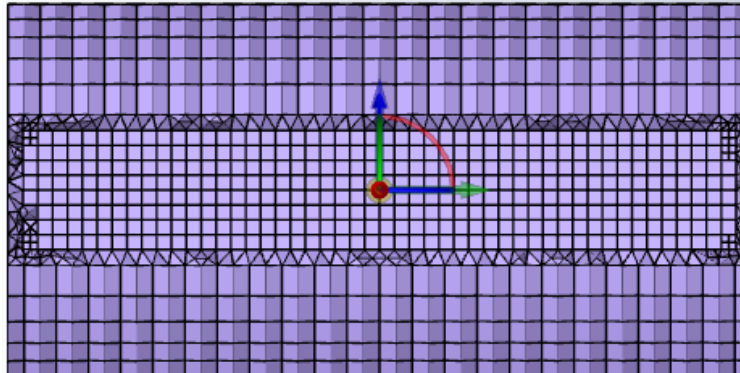
The mesh will appear as below, where the code will automatically create a Clipping Plane to help visualize the mesh inside the domain. Here the 5 layers of the boundary layer, the Hexcore mesh and the two Buffer layers are visible.



However, with the current setting the boundary layer will look different from what we are expecting. As shown above the layer does not grow continuously. This happens because the code automatically imposes some restrictions in case the boundary layer is too large for the current geometry.

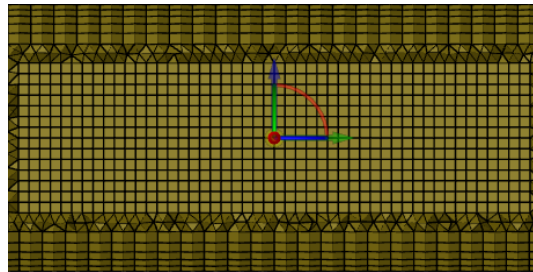
If you want to force the creation of the boundary layer with this setting, one workaround would be to modify the “Global Boundary Layer Setting” in “Generate the Volume Mesh”. First Click at the bottom “Revert and Edit” and then expand “Global Boundary Layer Setting” and change first the “Min Aspect Ratio” to 0 and the “Max Aspect Ratio” to 0. These values control the compression of the boundary layer. If you click “Update” the boundary layer will be generated as we expect.

Author	CFDLab@Energy	 Page 12 of 24
CFD for Nuclear Engineering	Session 2	




Effectively, this boundary layer is too large for the geometry we are considering. It could be useful to reconsider the size of the first cell and modify from 2mm to 1mm. To do so, select “uniform_1” in the Add Boundary Layers, click Revert and Edit and change the “First Height [mm]” from 2 to 1.

Click “Update” and “Generate the Volume Mesh”. The mesh should appear as below in the figure.



Note that if you reverted the values of Max Aspect Ratio and Min Aspect Ratio to the default values, 25 and 1, the mesh would be exactly the same. Hence, if the code judges that some parameter you have introduced are not appropriate, one way would be to “force” the code to do it in your way, or another way would be the modify your input to obtain a more reasonable grid.

Your first grid is completed!

Author	CFDLab@Energy		Page 13 of 24
CFD for Nuclear Engineering	Session 2		

4 Set-up the Case in Fluent

After Saving (Write -> Mesh -> CaseName), the mesh will be saved in msh.h5 format. To import the currently generated mesh to Fluent solver you can simply click “Switch to Solution” on the top-left of the Toolbar. The mesh will be automatically imported in meters, so there is no need to rescale.

4.1 Analyze the Problem

The problem is completely set by simply providing the Reynolds number ($Re = 500$) so it could be applied to any geometry and fluid. In a Periodic pipe the velocity cannot be imposed because the velocity profile will be the result of the simulation. We need to impose either the mass flow rate [kg/s] or the pressure drop. Once we know the Reynolds number it is straightforward to evaluate the mass flow rate, which would be:

$$\dot{m} = 3.51 \cdot 10^{-4} \left[\frac{kg}{s} \right]$$

4.2 Check the Mesh and Set-up Models


To set up the case please refer to the document “Lab_02-Laminar Elbow - Hands on session_CORRECTED.pdf” for those parts which were already explained in the previous laboratory.

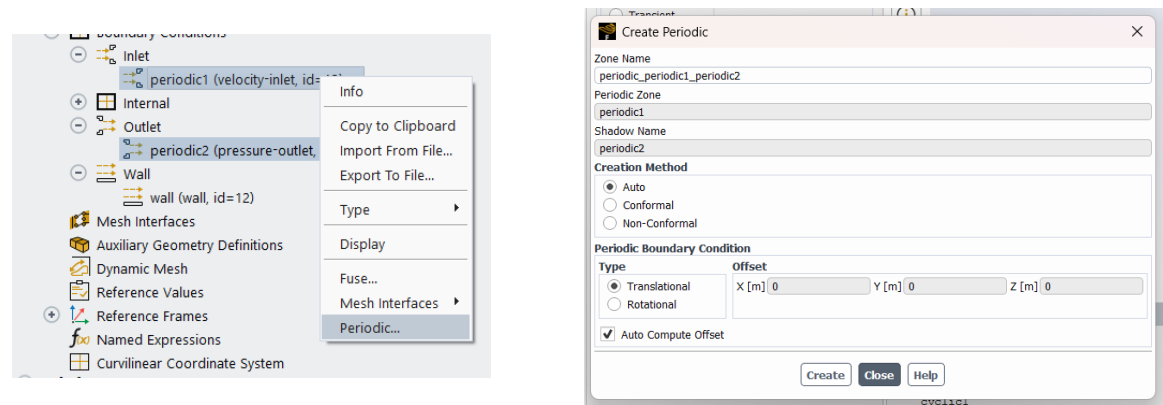
Please perform the following actions:

1. Check that the volumes and surfaces of the cells of the imported mesh are positive
2. Leave the default value of “Pressure-based” and “Steady”
3. Set-up the Viscous model as “Laminar”
4. Leave fluid as Air with the default properties

4.3 Define Periodic Boundary Conditions

We have set during the meshing process inlet and outlet conditions for the pipe faces where the fluid is supposed to flow. Hence, Fluent will automatically assign to Inlet and Outlet those boundaries. If you expand them, they will show the name Periodic1 and Periodic2. To define periodic boundary conditions simply select both of them using Ctrl, right click and select Periodic.

Author	CFDLab@Energy	 Page 14 of 24
CFD for Nuclear Engineering	Session 2	

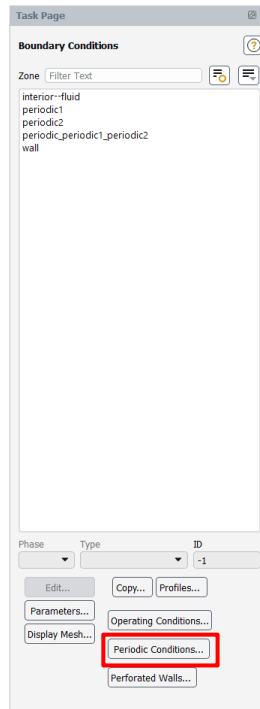


The Create Periodic window will pop-up in which the Periodic Zone and the Shadow Name are automatically selected. The creation method should be left Auto and the type is Translation.

Click Create.

The Inlet and Outlet node will disappear and the Interface and Periodic node will appear.

If you double-click on “Boundary Conditions” the Task Page will show all the boundaries and possible setting at the bottom. Click on “Periodic Conditions” as shown below




Here click on “Specify Mass Flow” and “Flow direction” according to the orientation of your domain. If you have oriented the domain as in this tutorial you need to change 1 from the X-direction to the Y-direction.

4.4 Define the Solutions

For this part also refer to the document “Hands-on Session 2” and set the model in the following way:

- Pressure-Velocity Coupling
 - Scheme: SIMPLE
- Spatial Discretization:
 - Gradient: Least Square Cell Based
 - Pressure: Presto!
 - Momentum: Second Order Upwind

Write -> Case

Author	CFDLab@Energy	 Page 16 of 24
CFD for Nuclear Engineering	Session 2	

4.5 Define Reports

We are going to define first a report on the mass flow rate to make sure that we have set everything correct. To do so double click on “Report Definitions”, New -> Flux Report -> Mass Flow rate


Change the name into “mass-flow-rate”, click on either Periodic1 or Periodic2 if you select Periodic_periodic1_periodic2 boundary the result will be zero, deselect “Report File” and click OK.

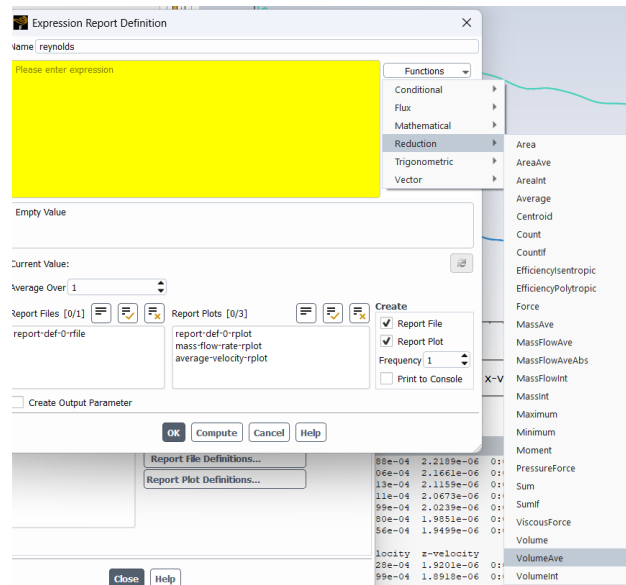
It would be important also to plot the average velocity at one of the boundaries to confirm that we reach the exact Reynolds number we are trying to impose. To do so please create an Area-Weighted Average by clicking New -> Surface Report -> Area-Weighted Average. Define the correct Field Variables, check the Option Per Surface, uncheck Report File and select one of the two periodic boundaries.

We could also create an expression to check the value of the Reynolds number directly. To do so select New -> Expression. Change the name into “reynolds-number”. The simplest way would be to write the expression of the Reynolds number with the properties and the diameter of the pipe as coefficients and only the average velocity computed by Fluent. However, it is often better, and surely more educative, to obtain all the data from the code so that the process would be general and we could select another fluid without the need to modify the report. To do so Fluent reports on the side the variables to be considered and some useful functions. The expression to be written is the following:

$$Re = \frac{\rho U D}{\mu}$$

The density can be found in Variables -> Density -> Density. However, this value is a Volume value, even if it is the same for the whole domain so it cannot be used to multiply the average velocity which is just a single value. In this case we need to “Reduce” it by volume averaging. Click “Functions -> Reduction -> VolumeAve

Author	CFDLab@Energy	 Page 17 of 24
CFD for Nuclear Engineering	Session 2	



The following script will appear in the expression

`VolumeAve(<expr>,[<location>, ...])`

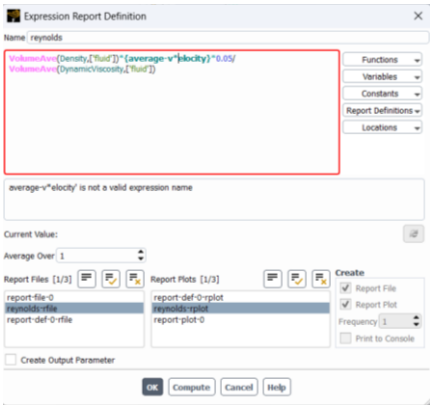
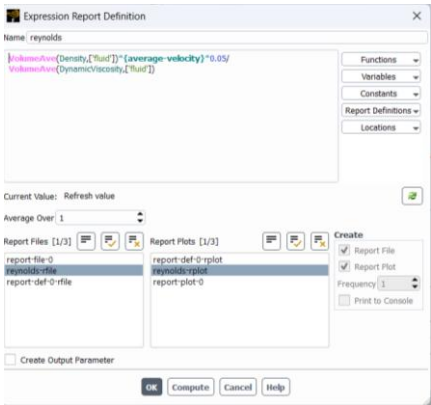
Select “<expr>” and click on Variables -> Density -> Density, the code will apply the proper terminology for the intended variable. For this case the variable name is intuitive but for others might be not, so it is always better to request the variable, unless confident on what variable to be introduced.

Select “<location>, ...” and click Locations -> Cell Zones and select Fluid.

Add the multiplication sing “*” and the average velocity with the Report Definition just done before, by clicking: Report Definitions -> average-velocity

Add the multiplication sing and add the size of the diameter. We could try to use a general procedure but the location will be the center cell location which might give a slightly but not negligible difference for the diameter. So for this case simply write 0.05.

Finally divide “/” by the VolumeAverage of the Dynamic Viscosity found in Properties as presented in the step for the density. If everything is done correctly the expression will appear like in the figure below on the left. If some syntax error is detected the expression area will be highlighted in red like in the figure below on the right.



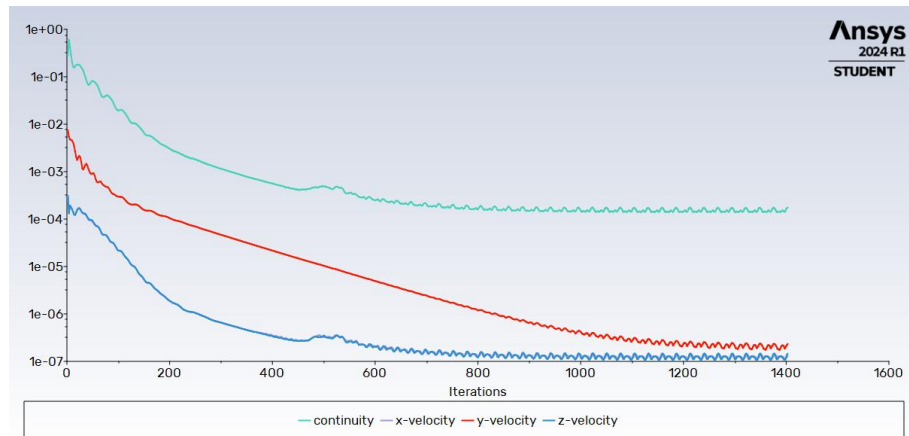
A hint on the error might be shown below.

4.6 Run the Simulation

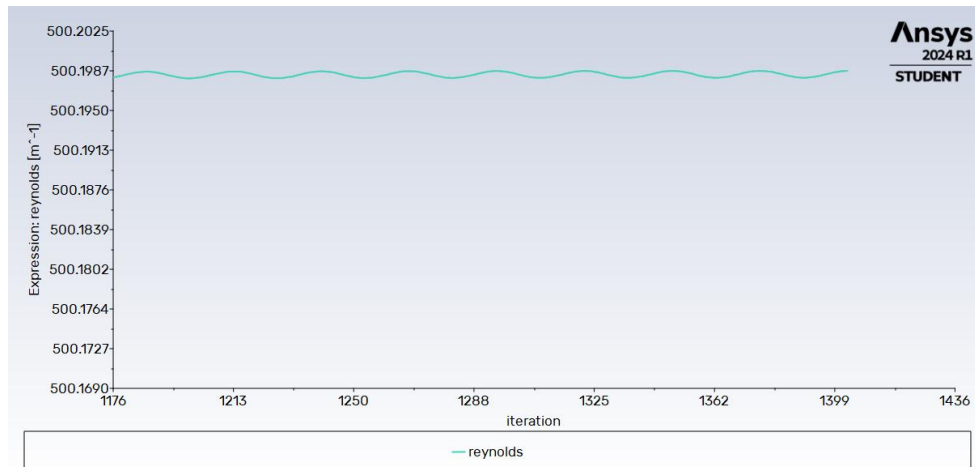
Before starting the computation Initialize the flow with the Hybrid initialization. Also uncheck the check on the convergence as presented in the Hands-on session 2. Run the simulation until you reach convergence judging the residuals as well as the reports.

4.7 Results

If the setup of the simulation was done in the same way as this tutorial the code should converge in about 1200 iterations as shown in the residuals below.




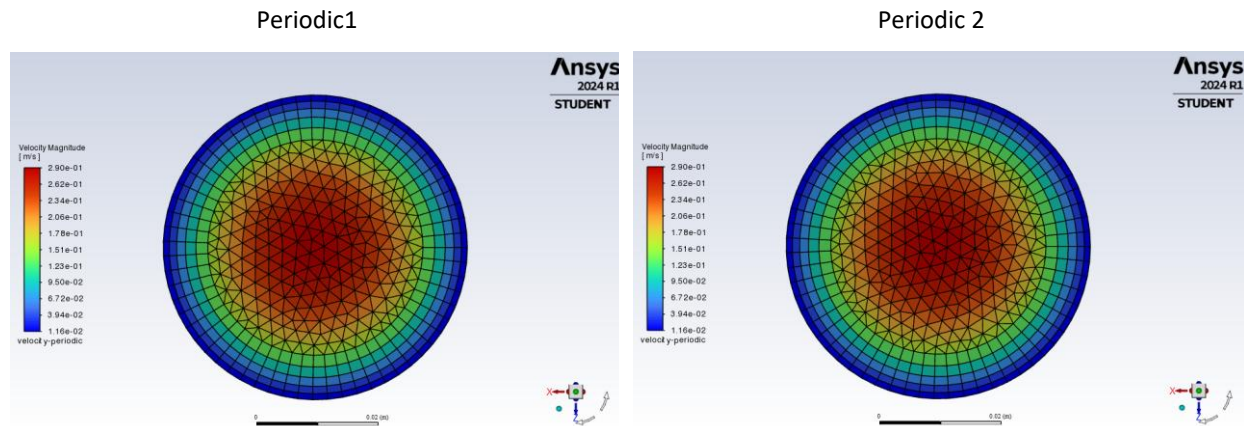
It is possible to confirm that the Reynolds number computed in the simulation is very close to the value we have imposed, i.e. 500.



4.7.1 Visualization of the Velocity Contours

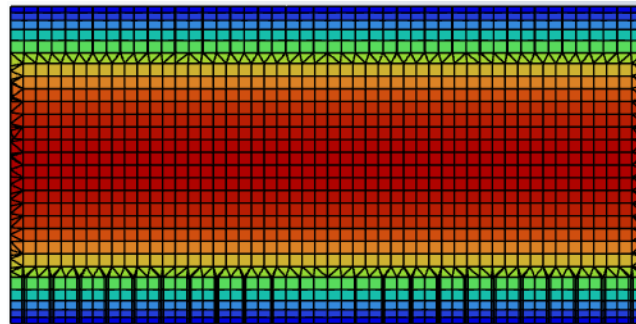
The first velocity contour can be visualized at one of the Periodic Boundary as below.

Author	CFDLab@Energy		Page 20 of 24
CFD for Nuclear Engineering	Session 2		



To change the color of the mesh to Black change the color of the Type “Interface”. We can notice that the velocity contour resembles what we expect from the flow we have imposed and that Periodic1 and Periodic2 which have exactly the same contour.

Try to show the contour on one plane passing in the middle of our domain parallel to the flow as below, creating a surface. Right click on Surfaces -> New -> Plane. Change the name to “midplane”, select XY plane and set the Z coordinate to 0. Click “Create”. Make a new Velocity Magnitude Contour and select the newly created “midplane” as surface, Save/Display. If the plane is not visible try to rotate the view. It should look like the image below. Change the grid color by changing the type “Surface”.




4.7.2 Compare the Velocity Profile

The velocity profile of a laminar flow can be solved analytically and equals to:

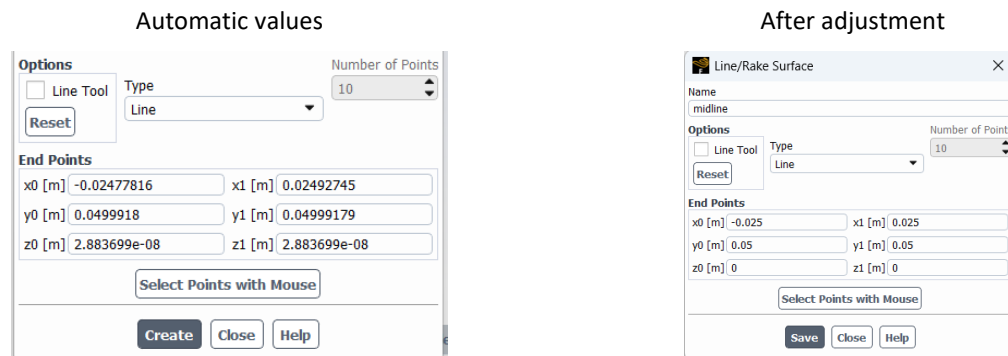
$$u(r) = 2\bar{u}\left(1 - \left(\frac{r}{R}\right)^2\right)$$

Where \bar{u} is the average geometry. R is the radius of the pipe.

Author	CFDLab@Energy		Page 21 of 24
CFD for Nuclear Engineering	Session 2		

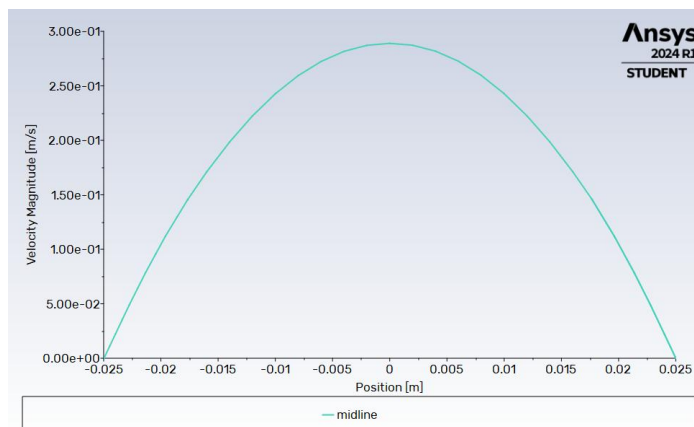
To extract the velocity profile first we need to create a line where we want to extract the velocity. Leave the midplane contour open and right click Surface -> New -> Line/Rake. Change the name to “midline”. We want to locate the line in the middle of the plane for convenience. If we know the coordinates of the initial and end point of the line we could simply type those coordinates. In case we are not sure we could use a graphical method to help us to remember the exact location. Press “Select Points with Mouse”. A message will appear asking you to use the “MOUSE-PROBE mouse button”, which is general is the right click of your mouse. So do not close the message and click as close as possible to the two nodes where you would like to place the line. The coordinates will adjust automatically.

It could be convenient to adjust the values to round up as in the example on the right side of the image below.




Click Create

Finally create an XYplot with the method already presented in the Hands-on Session 2 and show the velocity profile on the midline as below.



We need to specify in which direction to plot the Magnitude Velocity. In this case in the X-direction.

Author	CFDLab@Energy		Page 22 of 24
CFD for Nuclear Engineering	Session 2		

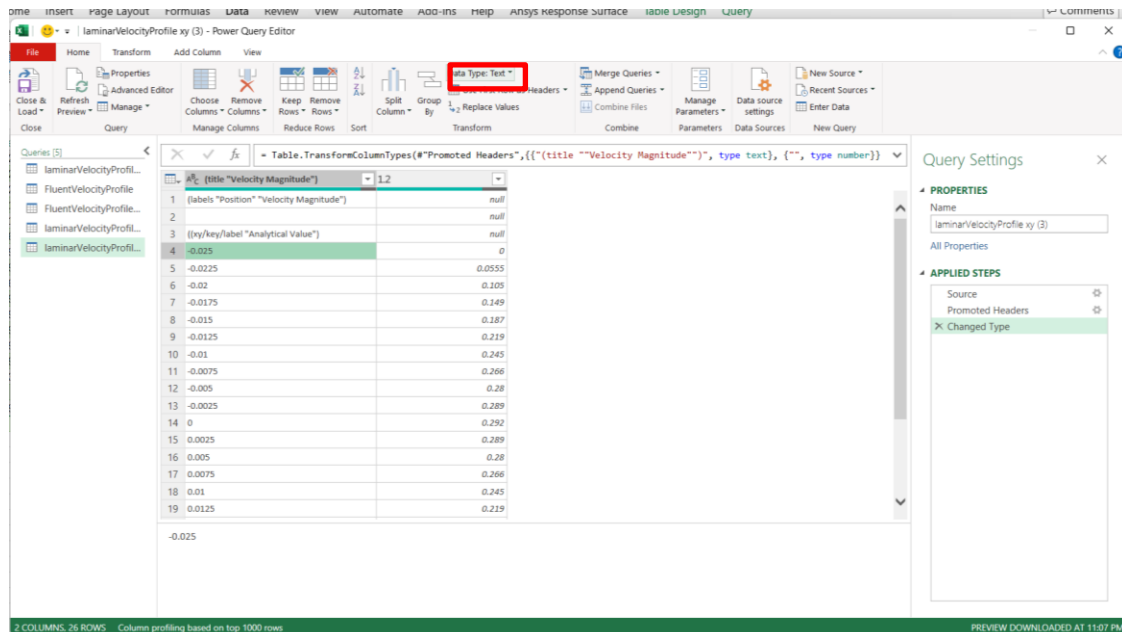
4.7.3 Export the Velocity Profile for Comparison

Often it is important to compare the simulations results with analytical or experimental results for validation. We can achieve this in two ways. One exporting the CFD results and analyzing them with external software, such as MATLAB or Excel, or vice versa importing the analytical/experimental results into Fluent. The first one is straight forward and will be presented here.

On the XY Plot select “Write to File” and “Order Points”. Select “midline” and the direction where the line is oriented. Then click “Write...”. The file will be created as .xy extension.

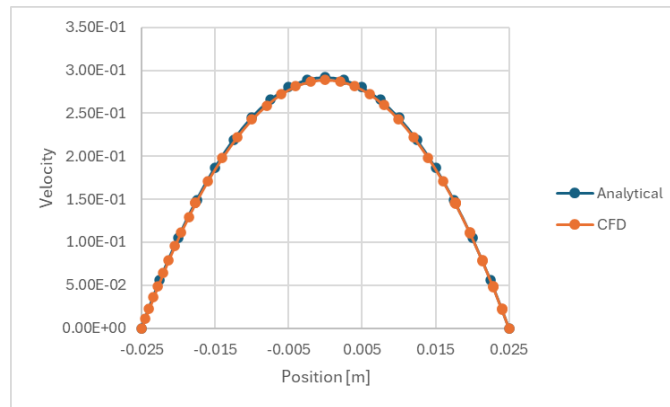
You can import the file with Excel by opening a new Excel document, navigate to “Data” and click on Text/CSV. After importing the data, you can plot the CFD data together with the analytical data.

Tip! For some reason the first column of the .xy file is imported as text so you will have a hard time to plot it correctly. Hence, when you import the data click on “Transform Data”. A new window will open. Select the first column and change the Data Type from Text to Decimal Number. Click on “Replace Current”.



Close and Keep the changes. Now you can use your data for plotting.

The result of the comparison should look like the plot below.



5 Compare two Different Grids Type

Now that we have completed this tutorial you have all the tools you need to run the same simulation but with a different mesh style. You are requested to simulate the same fully developed pipe flow but with a tetrahedral mesh and the same boundary layer. You are requested to:

1. Compare the number of elements
2. Compare skewness, orthogonal quality by means of histogram and contours.
3. Compare the number of iterations it takes to achieve convergence. Which mesh achieves faster convergence?
4. Compare the level of convergence. Which mesh reaches a lower value of the residuals?
5. Compare the velocity profile. Which mesh has a velocity profile closer to the analytical value? Can you quantify the error?