Computational Fluid Dynamic Analysis of a Laminar Flow Using Axisymmetric Technique using ANSYS R2 Student Edition

Author: Cyrus Hagigat, Ph.D., PE

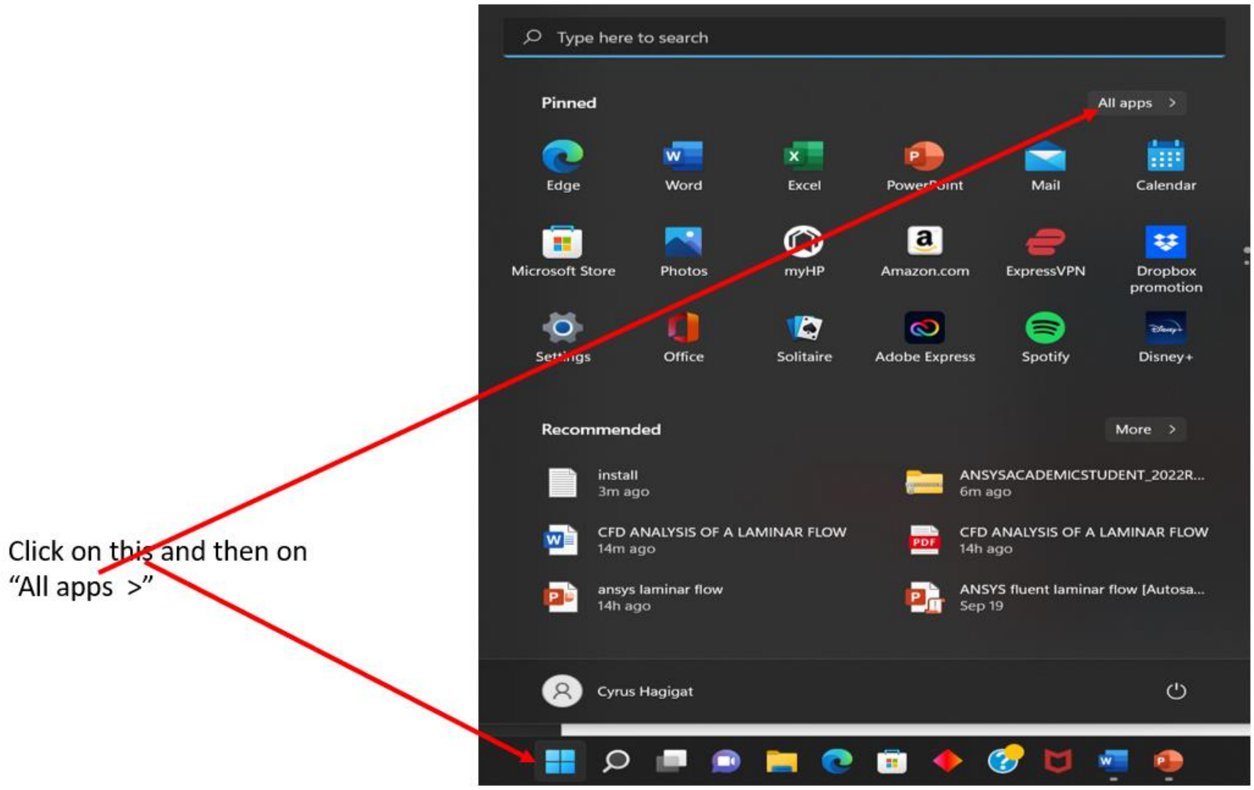
I: Introduction

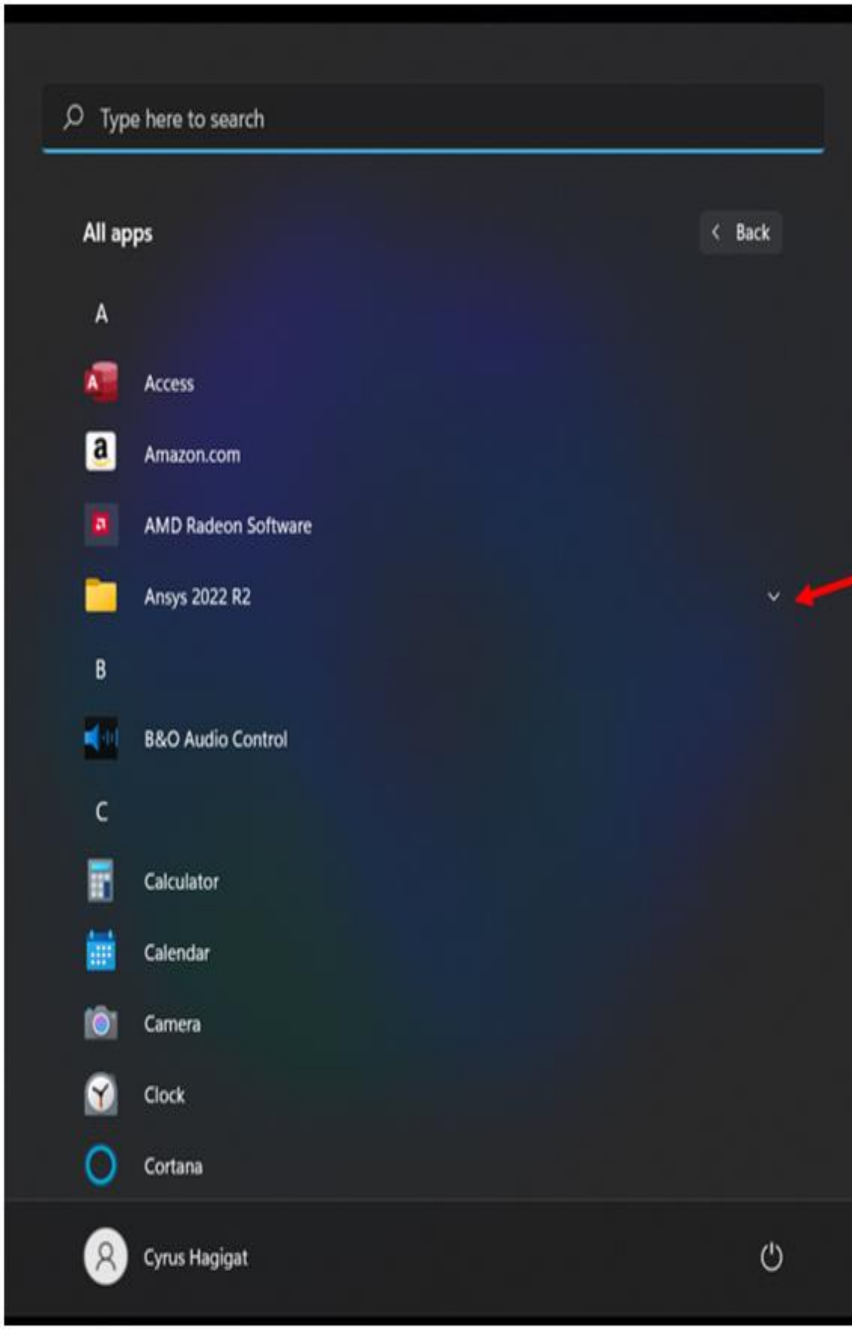
ANSYS is a multi-functioning Finite Element based software that can be used for modeling physical systems. ANSYS can be used for traditional and original uses of the Finite Element Analysis (FEA) such as stress and vibration analysis. ANSYS can also be used for analyzing fluid and gas flow using the Computational Fluid Dynamics (CFD) technique.

In this article, ANSYS is used for analyzing the laminar flow of fluid in a pipe and graphically displaying the fluid distribution in the pipe. The simulation was used in a Fluid Mechanics classroom in the author's institution and the results compared against experiments performed in a fluids lab.

This article presents in great detail every step of using the latest version of the software as of the date of this writing. As new editions of the software are inevitably released, there will be minor differences in the Graphical User Interface (GUI) of the later versions of the software and this article. However, the main concepts will remain the same.

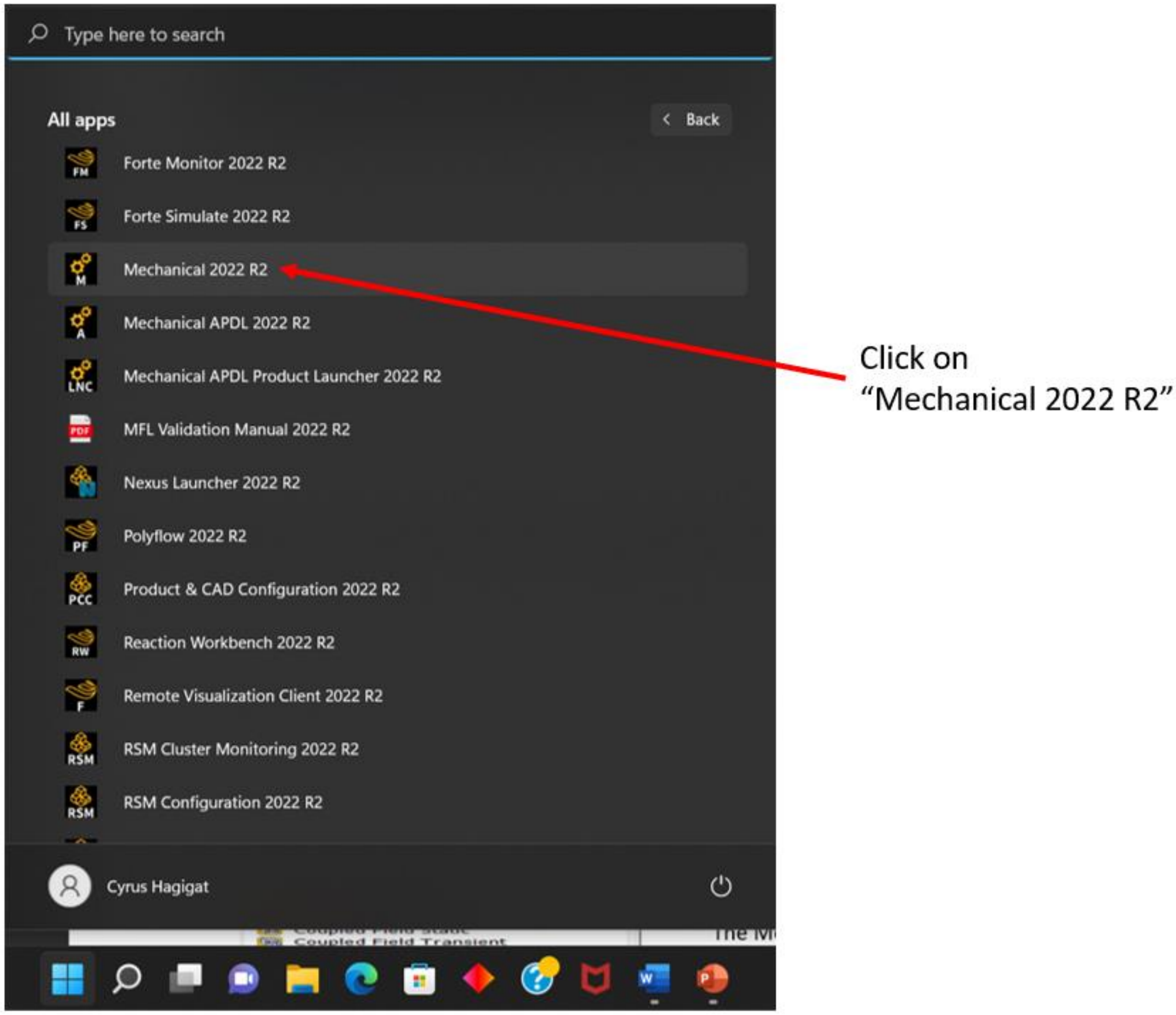
II: Detail description of the software interface and the implementation of the software to model fluid

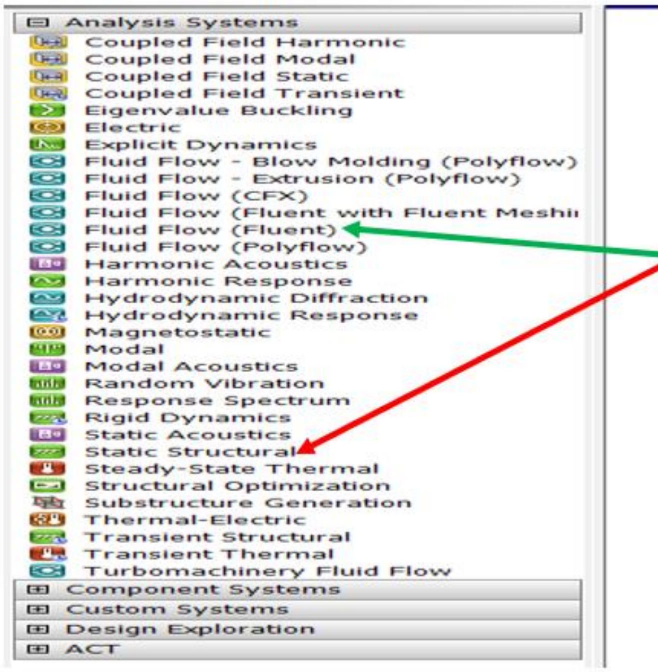




Click on the arrow next to Ansys 2022 R2

After clicking on the arrow above, the ANSYS modules appear. Go down the list and choose on "Mechanical 2022 R2".

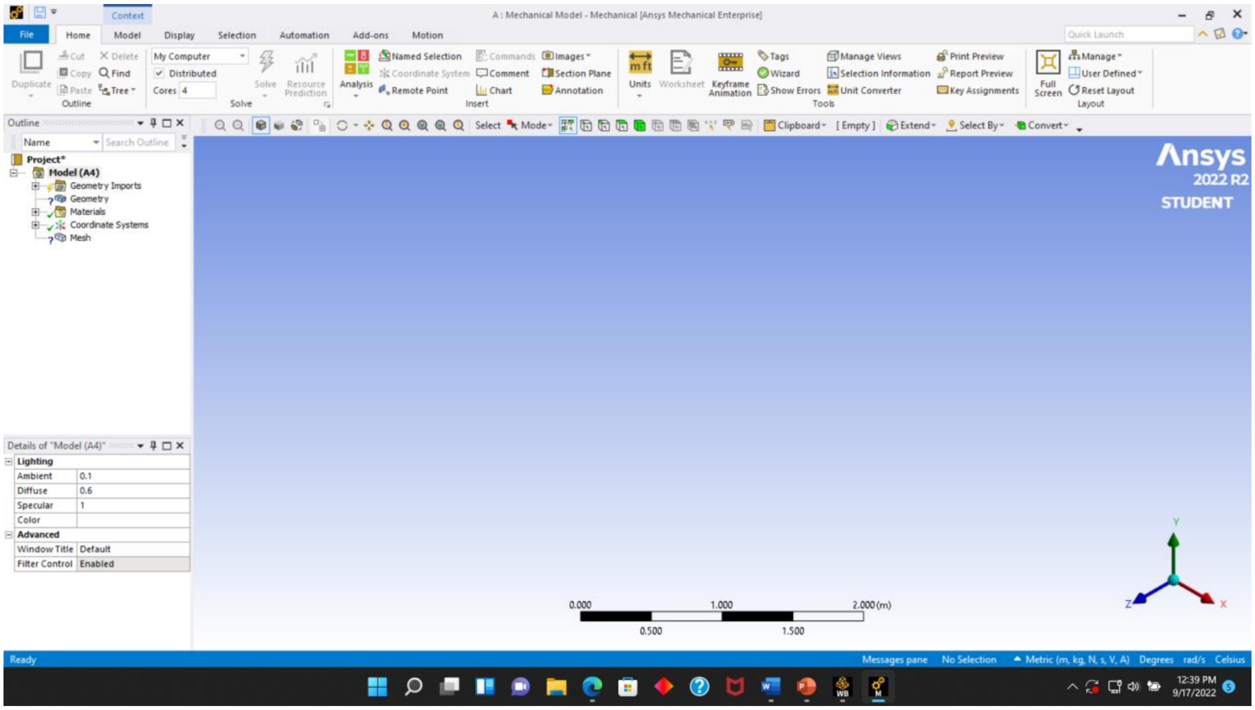




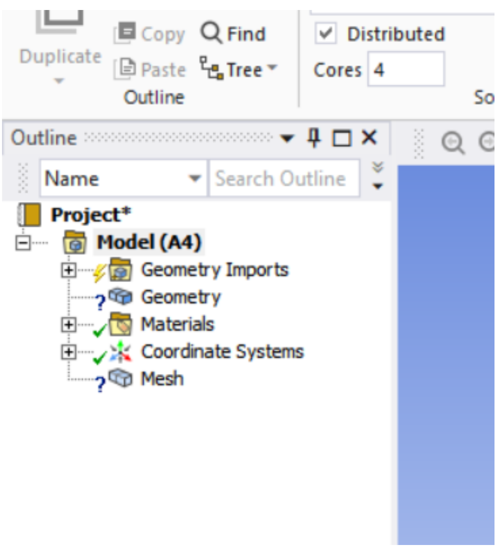
The Mechanical Module contains numerous portions as shown on the left. Some of the more familiar ones are "Static Structural" and "Fluid Flow (Fluent)"

By not clicking on the any of the modules, the software automatically (after a while) brings up a "DM" window that is generic and not related to any specific analysis type such as fluids or static stress analysis. "DM" stands for "Design Modeler". "DM" is an icon at the bottom of windows menu.

The generic “DM” screen appears as shown below.



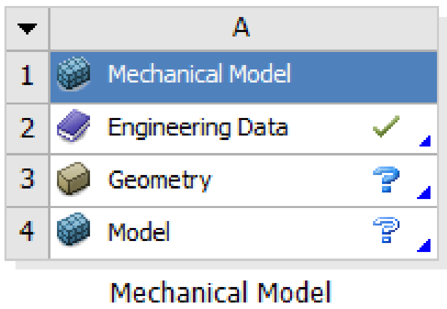
The following is the left side of above zoomed in.



DM stands for Design Model. There is a “DM” icon at the bottom of the screen. By clicking on it, the DM screen will appear and disappear.

At this point, in addition to the "DM" screen, there is a "WB" screen at the bottom. "WB" stands for "Work Bench". DM screen is defined as M at the bottom of windows screen.

As a result of the automatic addition of the "DM" screen, the following is added to the "WB" screen.

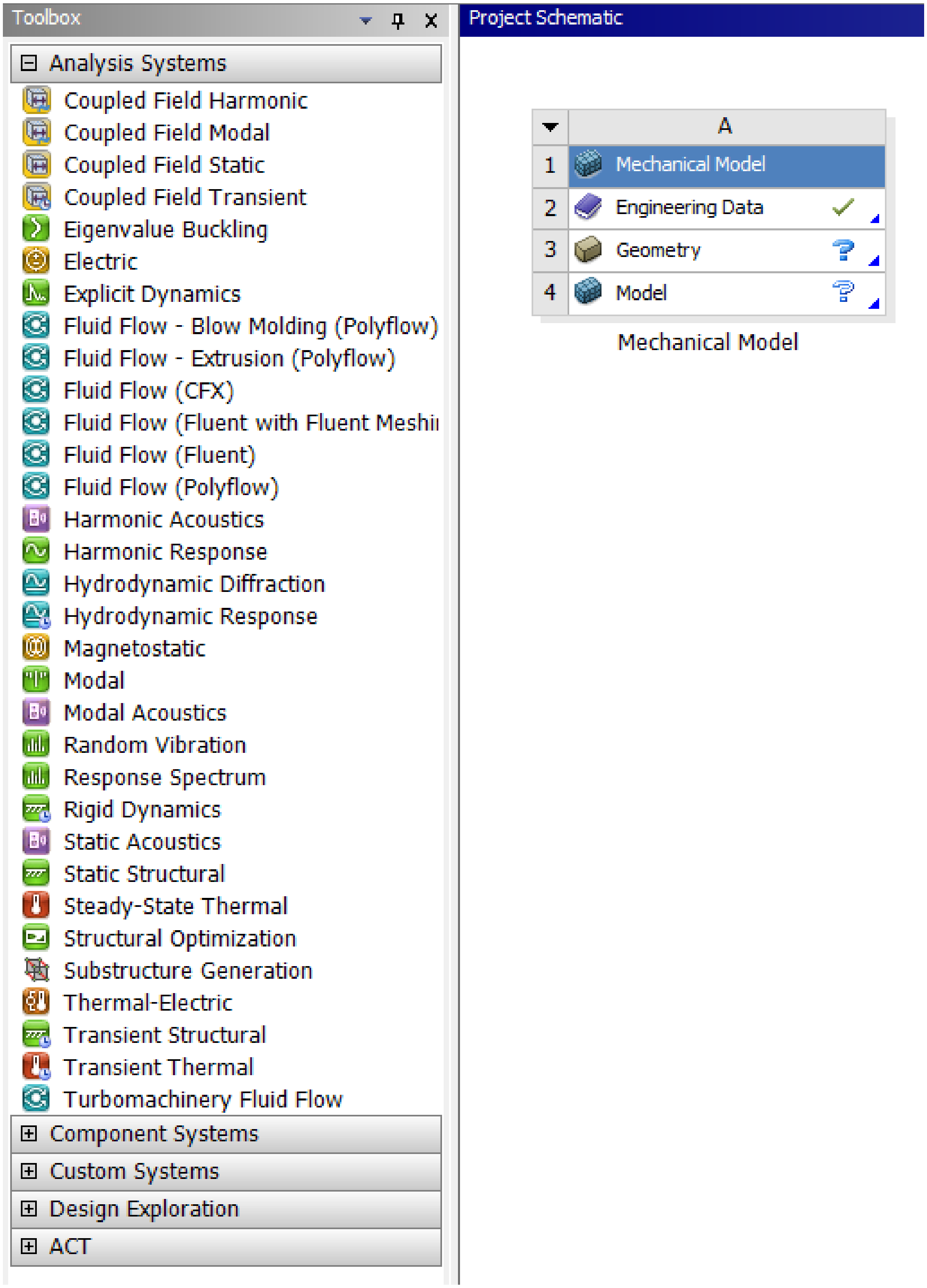


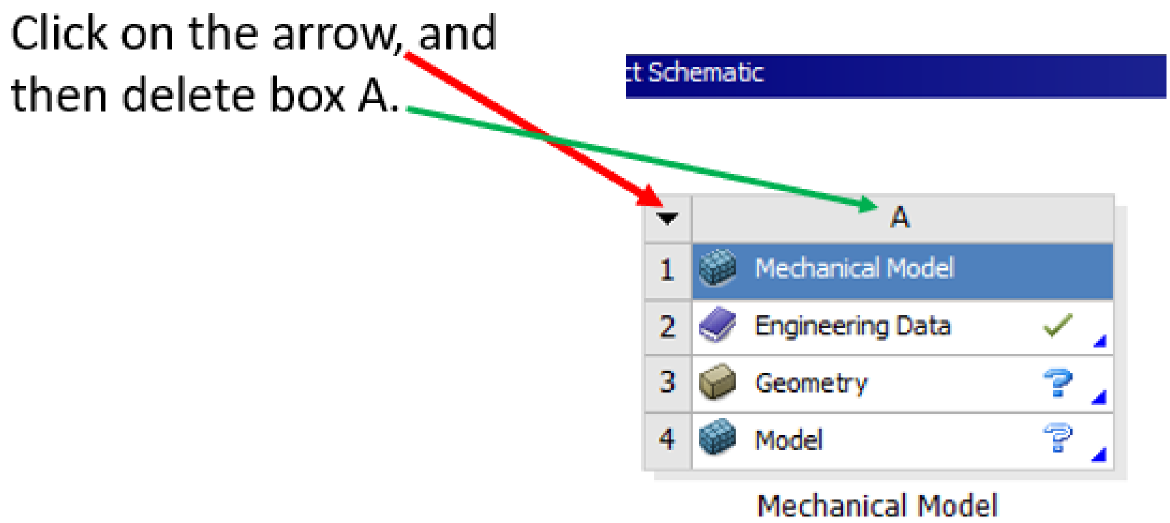
Note that the above indicates a "Mechanical Model" because the Mechanical module of ANSYS was used to start the process.

Close the "DM" window by clicking on the X on its upper right corner.

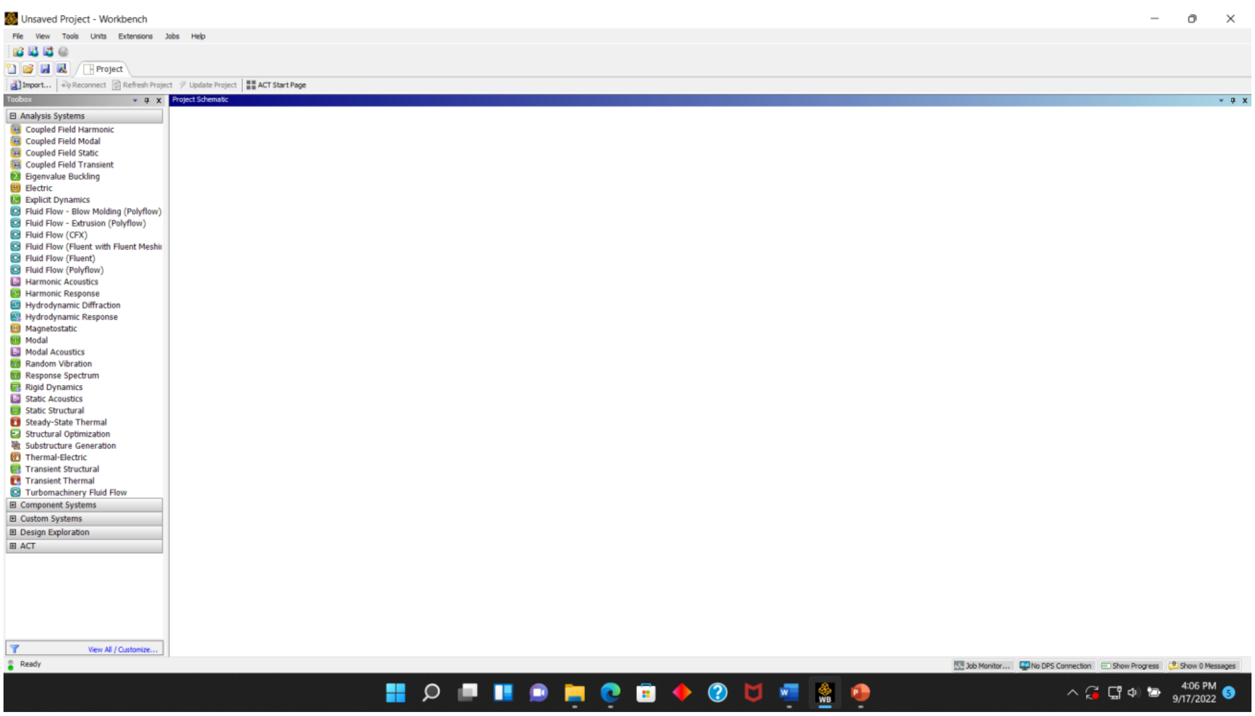
Bring up the "WB" window again.

The screen appears as shown below.

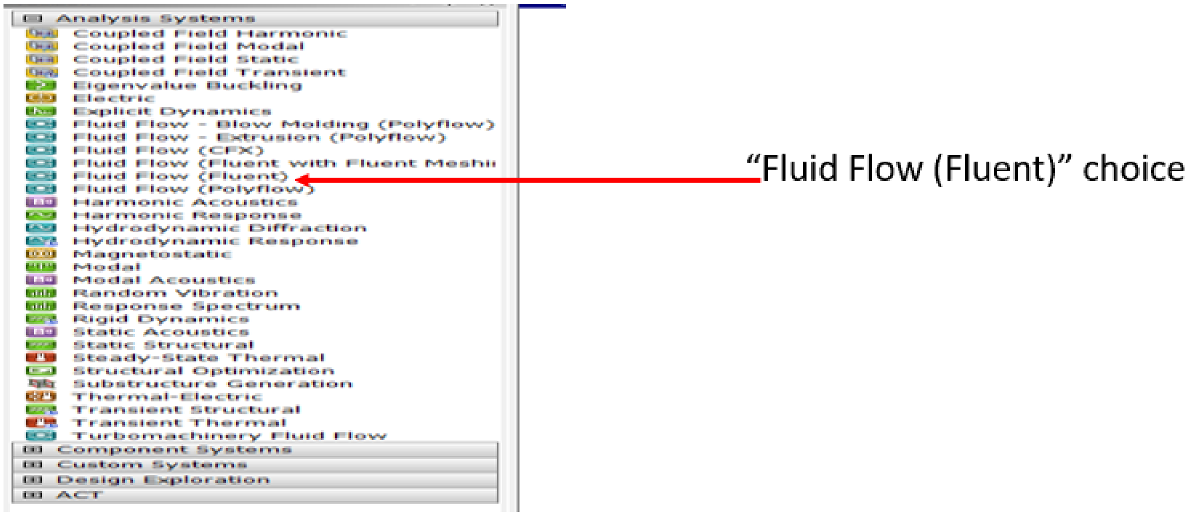




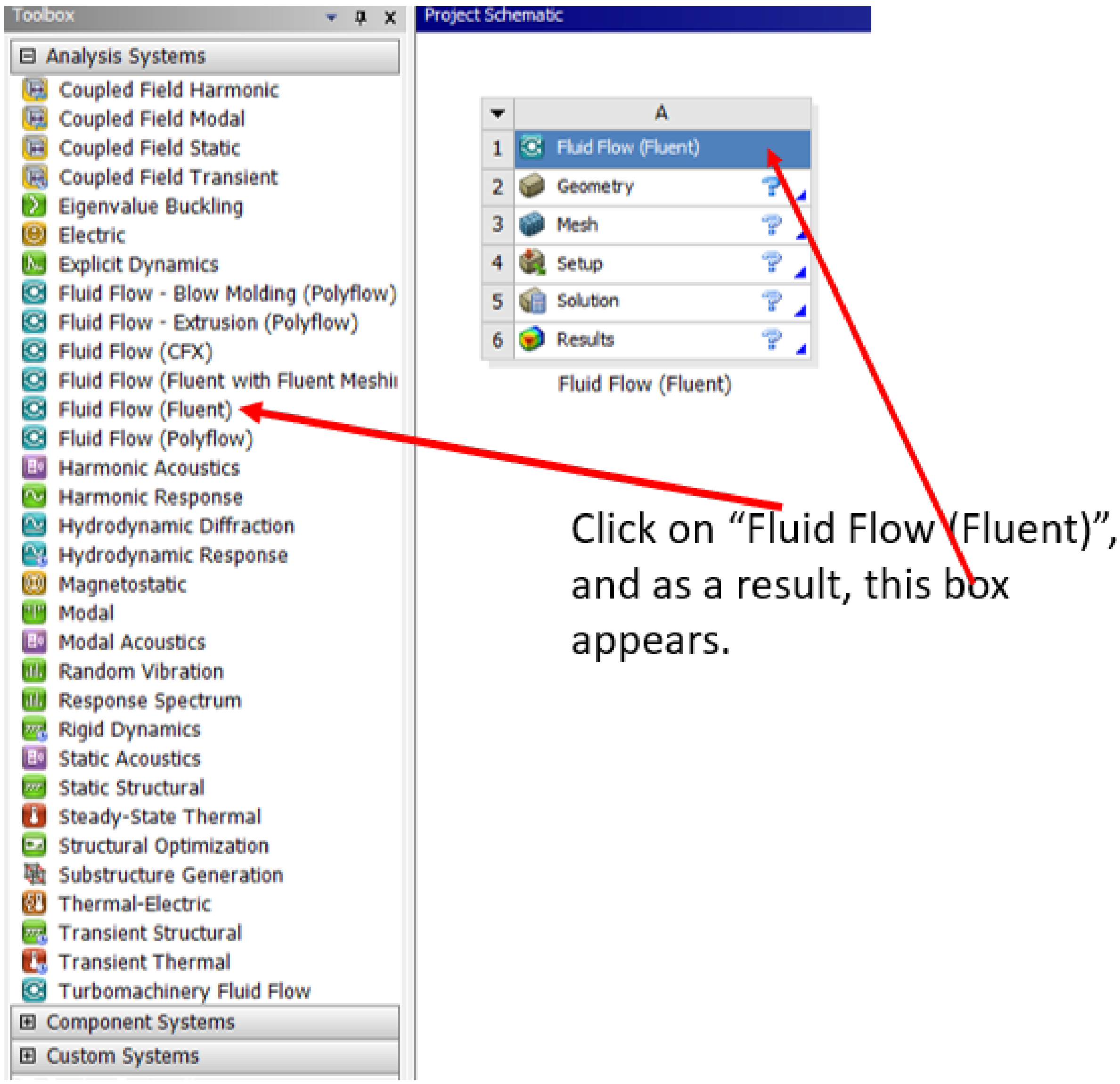
The screen appears empty as shown below.



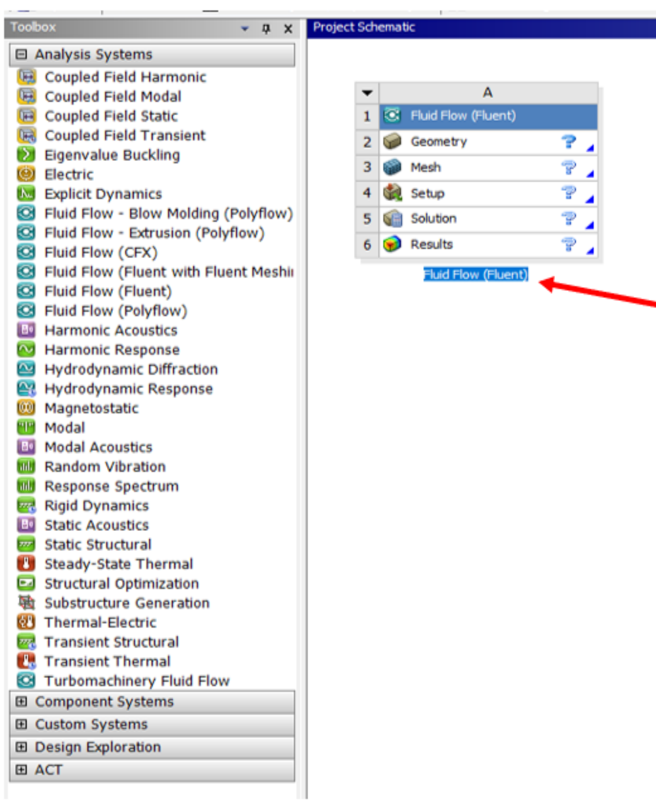
Click on “Fluid Flow (Fluent)” on the left.



Instead of clicking on “Fluid Flow (Fluent)”, the “Fluid Flow (Fluent)” can be dragged, and the screen appears as shown below.

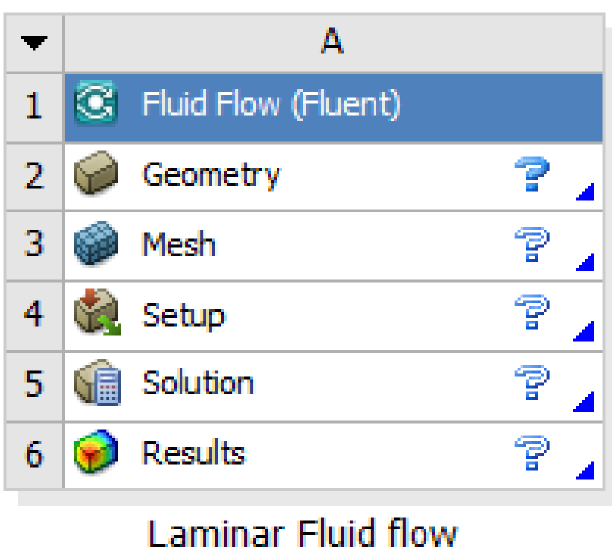


Alternatively, click on "Fluid Flow (Fluent)" and drag it, and the screen will appear as shown above.



Double click here.
It becomes blue highlighted as shown.
Once it is highlighted, a new name can be type in. In this case the name is changed to “laminar fluid flow”.

After the name is changed, the box appears as shown below.



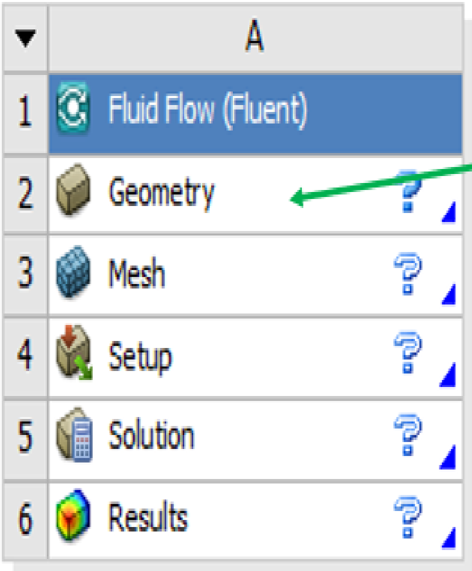
Click on "Geometry" above.

On the right of the "WB" screen, the following appears.

Properties of Schematic A2: Geometry

	A	B
1	Property	Value
2	⊟ General	
3	Component ID	Geometry
4	Directory Name	FFF
5	⊟ Notes	
6	Notes	
7	⊟ Used Licenses	
8	Last Update Used Licenses	
9	⊟ Basic Geometry Options	Used Licenses
10	Solid Bodies	☑
11	Surface Bodies	☑
12	Line Bodies	☐
13	Parameters	Independent
14	Parameter Key	ANS;DS
15	Attributes	☐
16	Named Selections	☐
17	Material Properties	☐
18	⊟ Advanced Geometry Options	
19	Analysis Type	3D
20	Use Associativity	☑
21	Import Coordinate Systems	☐
22	Import Work Points	☐
23	Reader Mode Saves Updated File	☐
24	Import Using Instances	☑
25	Smart CAD Update	☑
26	Compare Parts On Update	No
27	Enclosure and Symmetry Processing	☑
28	Decompose Disjoint Geometry	☑
29	Clean Geometry On Import	☐
30	Stitch Surfaces On Import	None
31	Mixed Import Resolution	None
32	Import Facet Quality	Source

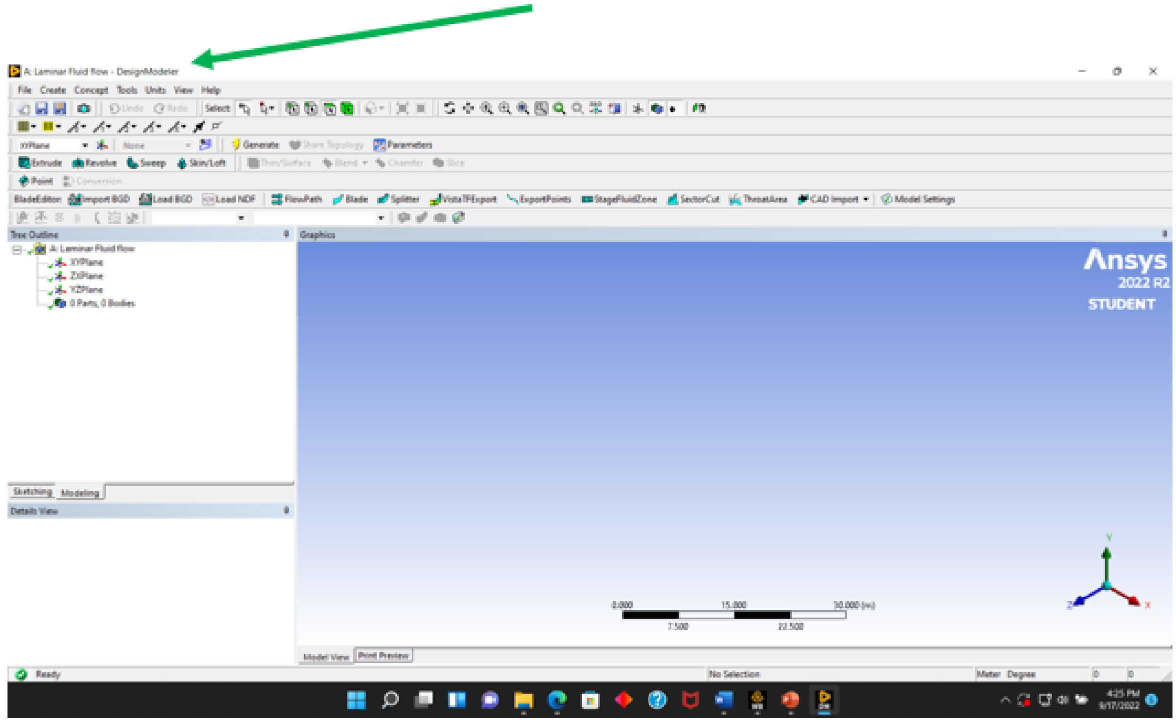
Click on this arrow.
Choose "2D" as the option



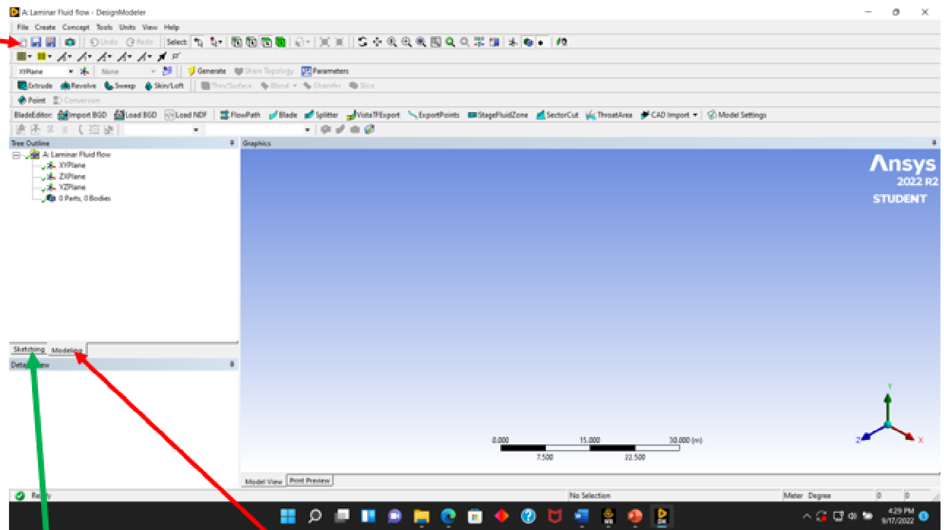
Laminar Fluid flow

Doble click on "Geometry".
A second screen tilted
A: Laminar Fluid flow – Design Modeler appears. This screen is defined as "DM" at the screen bottom

It cannot be seen. But, this is
A: Laminar Fluid Flow – DesignModeler



This undoes every things and sets the screen to its original form as show to the right.



This is the “Sketching” option

This is the “Modeling” option

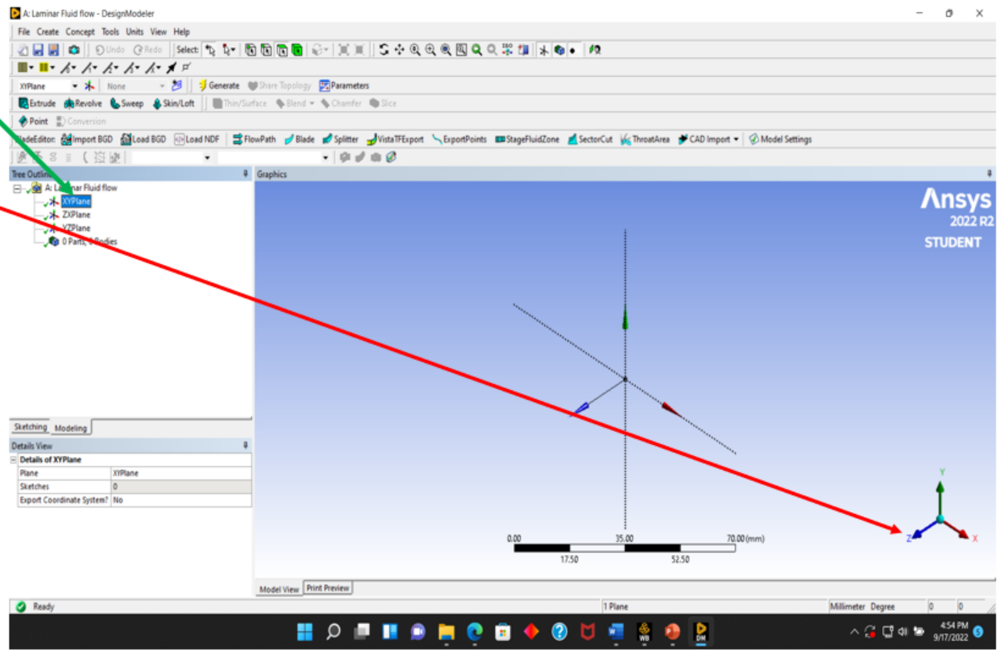
The following is the fluid flow problem statement.

Problem statement:

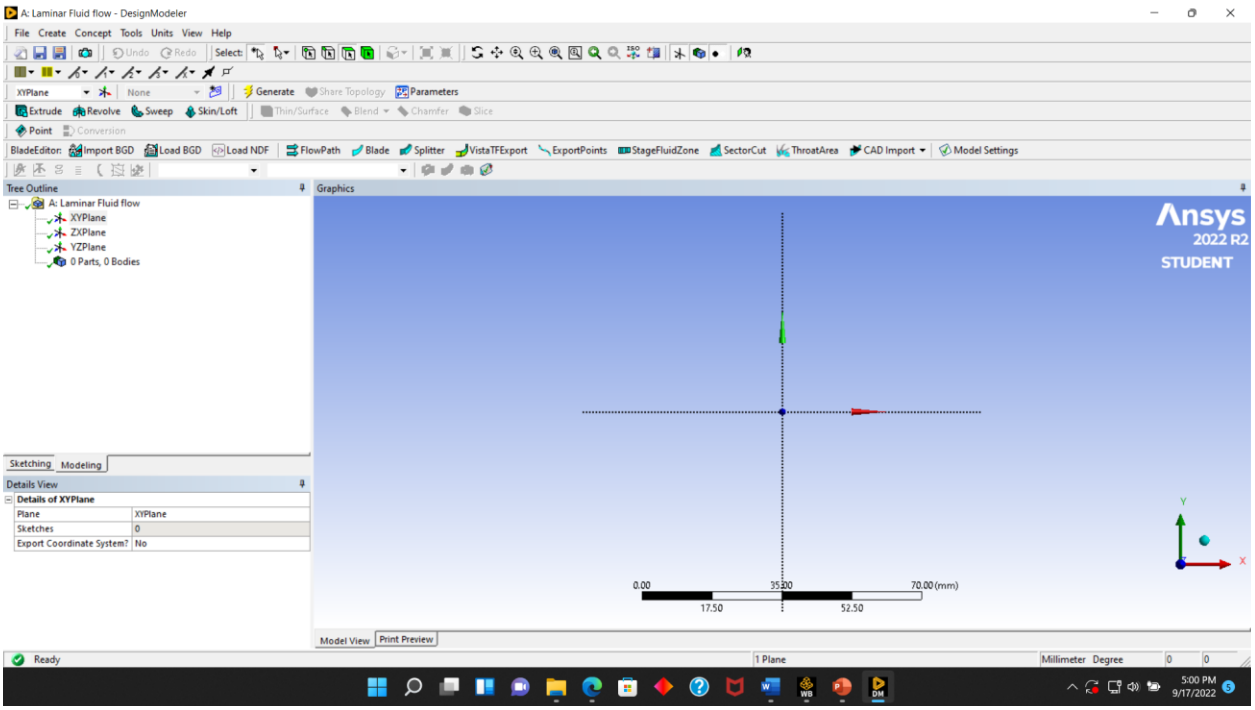
It is a 10 mm diameter pipe, its length is 100 mm. The water is entering into it, at a velocity of 0.0005 m/s. You need to find the velocity variation in the pipe.

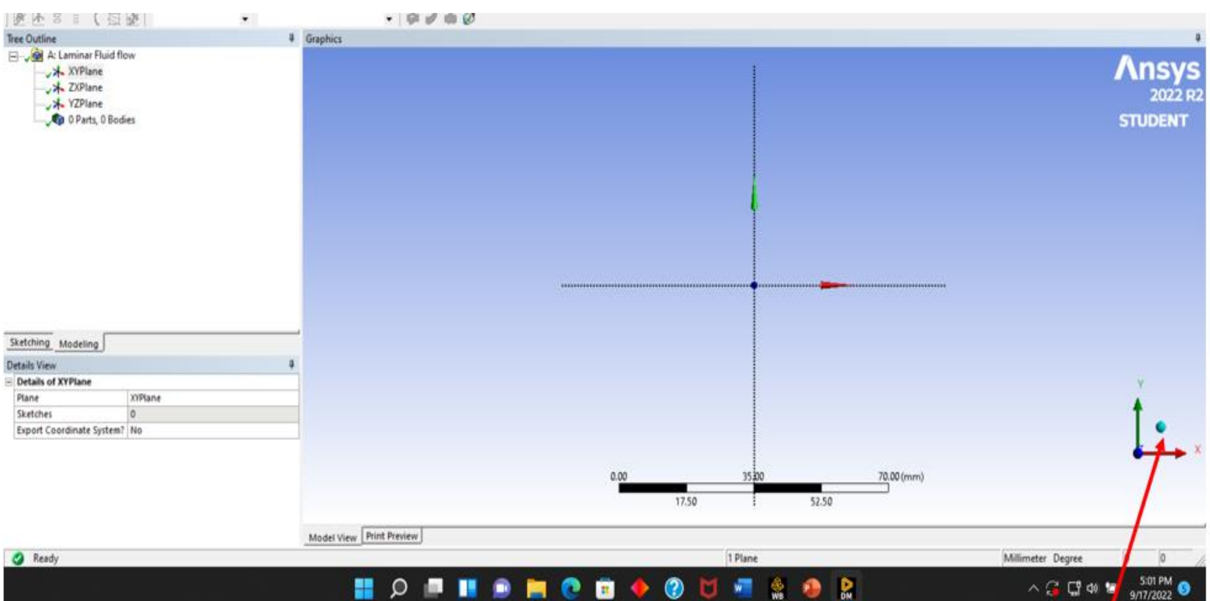
In the “DM” window, click on Units on top and change the default units to millimeter.

Click on “XYPlane”
Click On Z to make the Z axis normal to the displayed plane

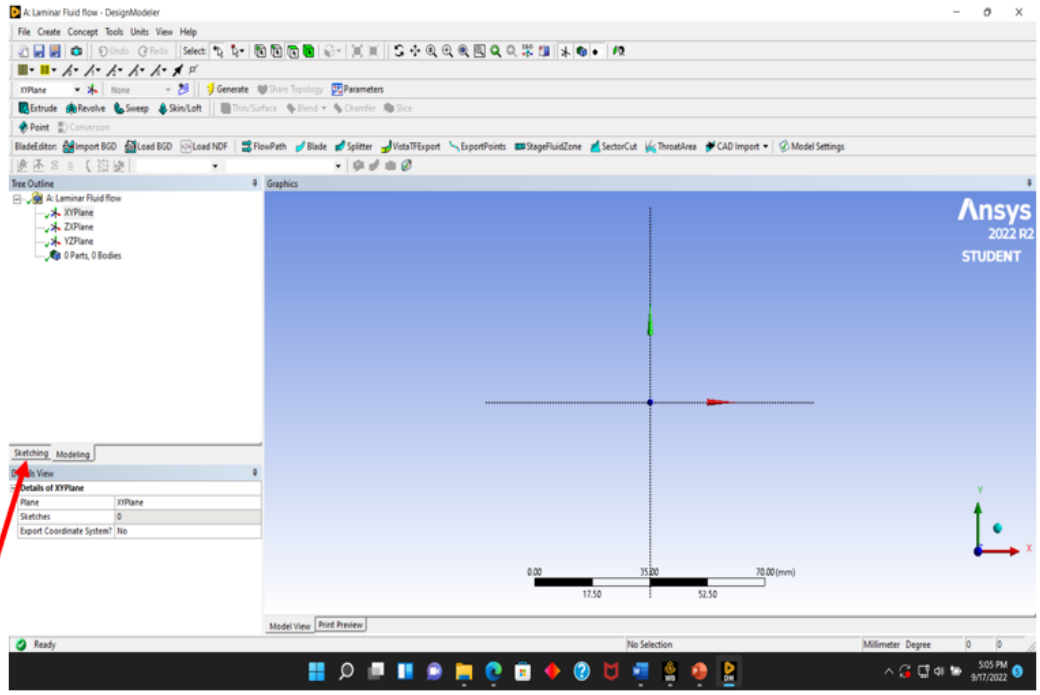


The screen with XY plane and Z normal to the plane appears as shown below.



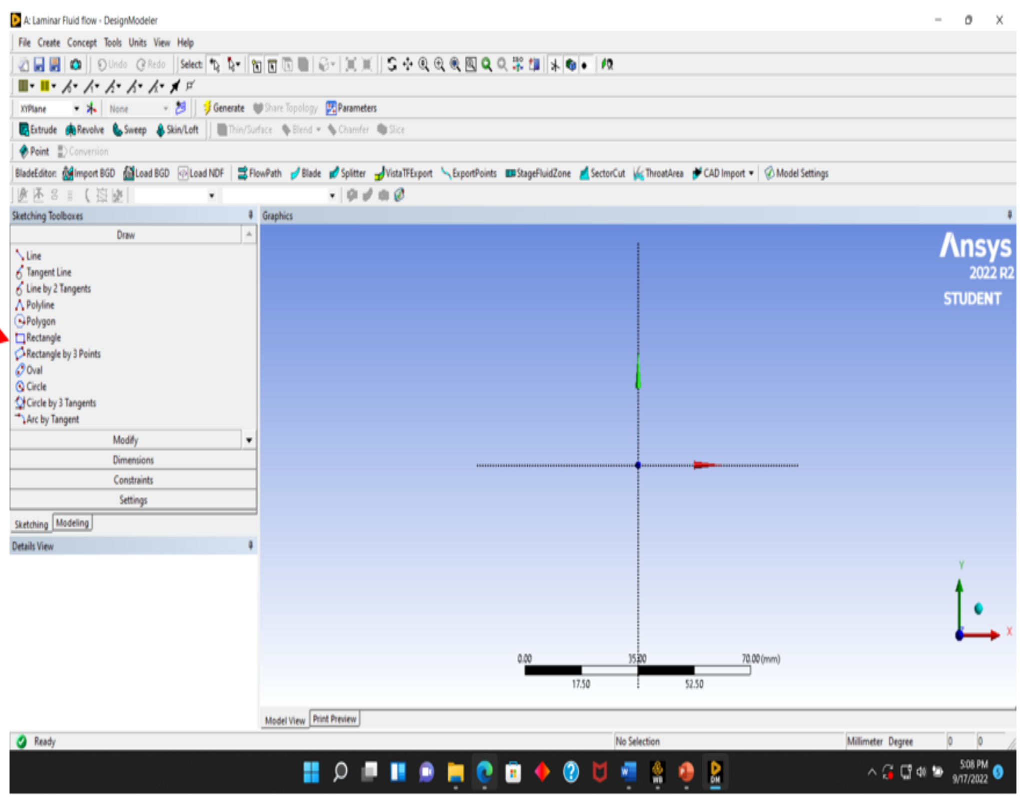


Click on this ball to put the coordinate system to the original default

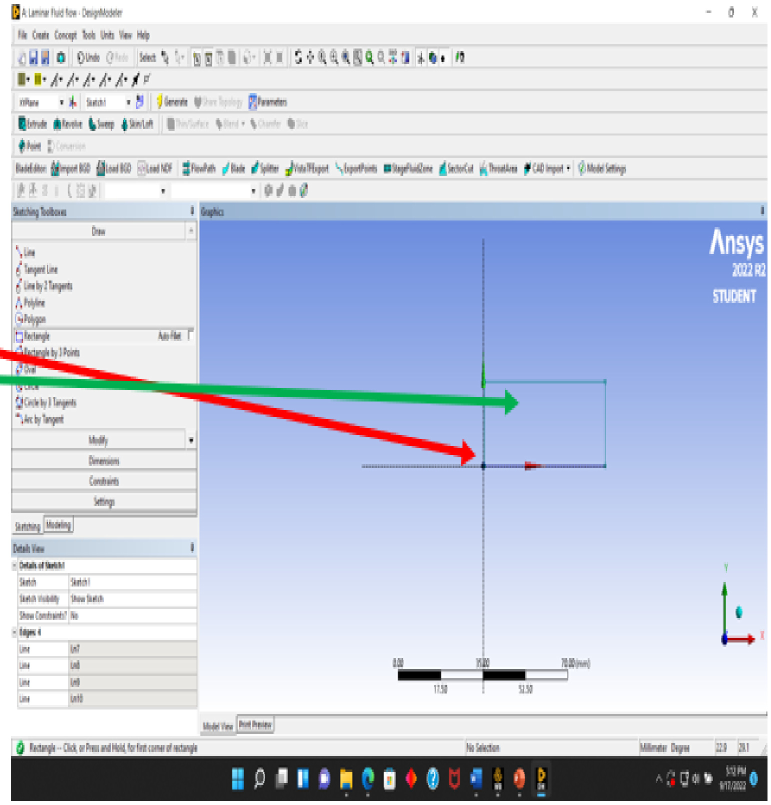


Select the “Sketching” tab

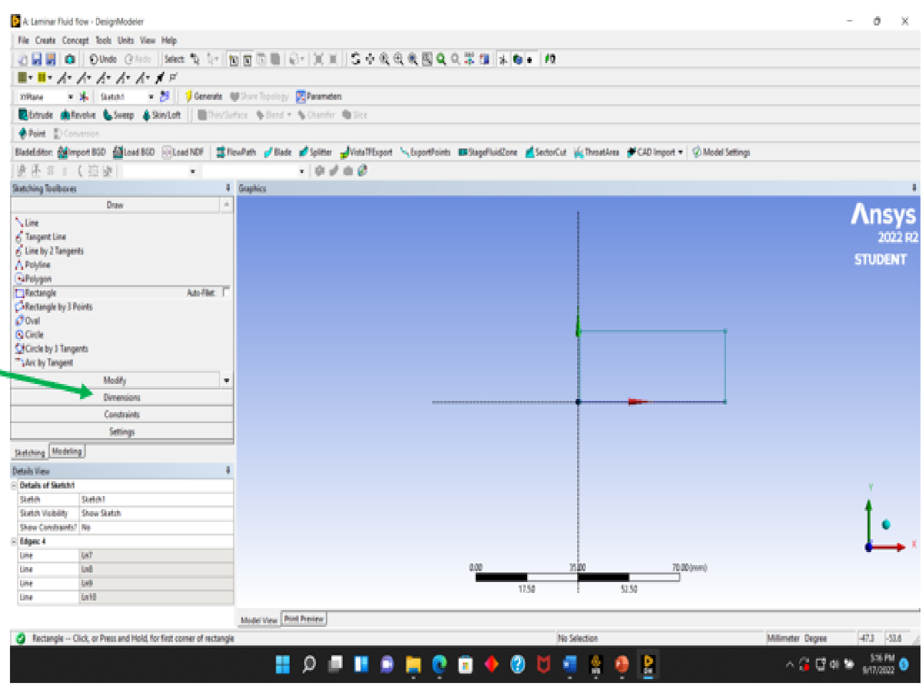
Click on “Rectangle”



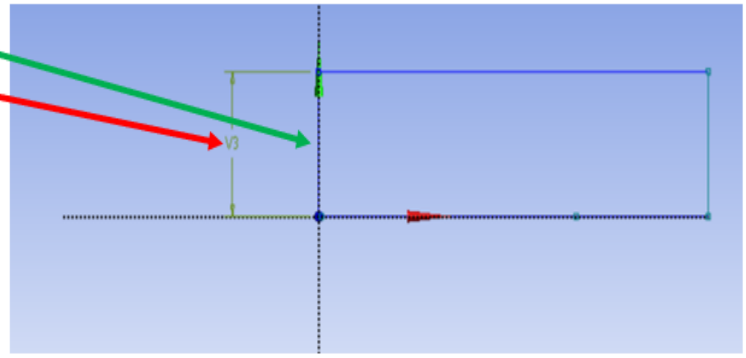
After clicking on ‘rectangle” as called for on the previous page, the pointer becomes a pencil. Click on “origin” and drag and create a rectangle



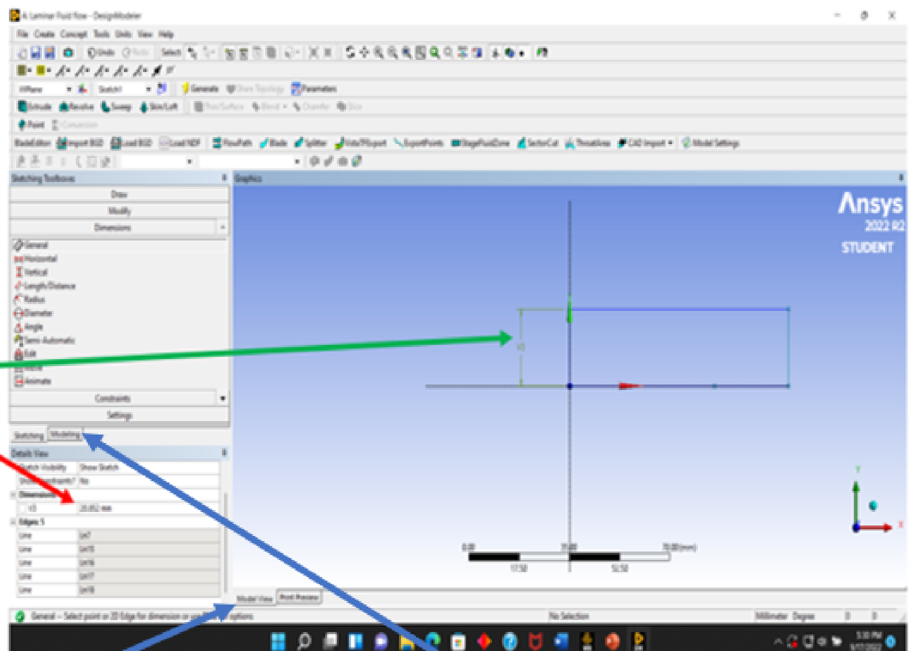
Click on “Dimension”



After clicking on “Dimension”, click on this side, and then drag and let go. The Dimension appears as shown.

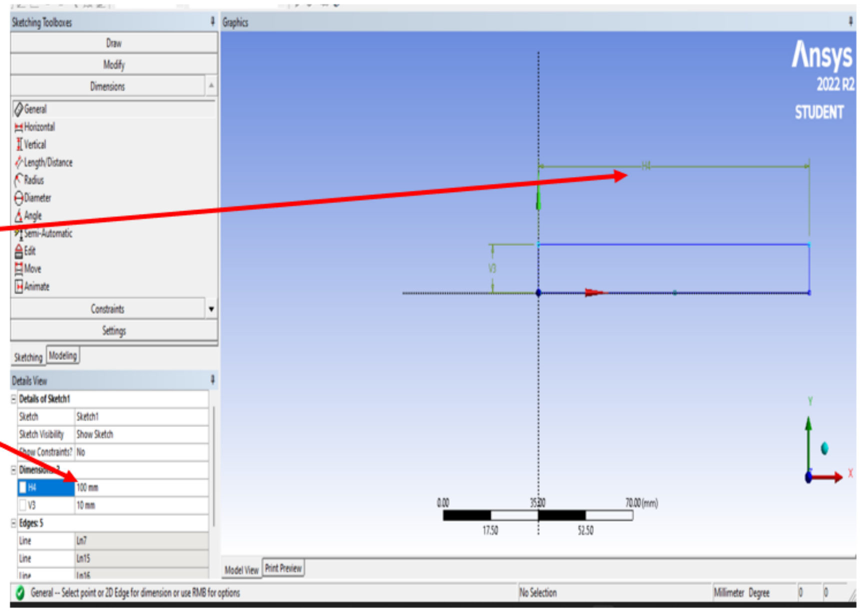


The dimension for “V3” is 20.852 mm. This can be changed to 10 mm by clicking in the box and changing the dimension.



If at any point, the menu on the “DM” screen does not look right, Click on “Modeling” and then “Sketching”, and the correct menu will pop up

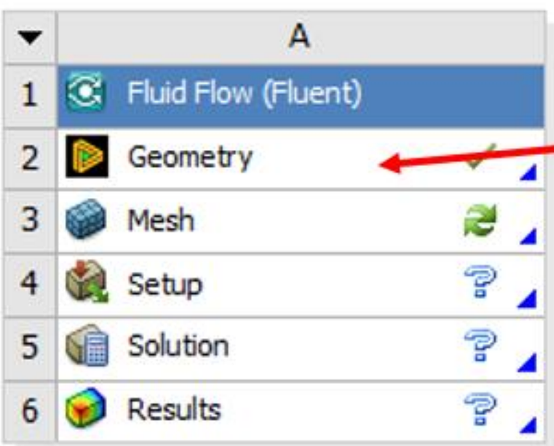
Using a similar procedure for dimension V3, create dimension “H4” and set it to “100”



At this point, the sketch is complete.

The sketch needs to be converted into a 2D geometry.

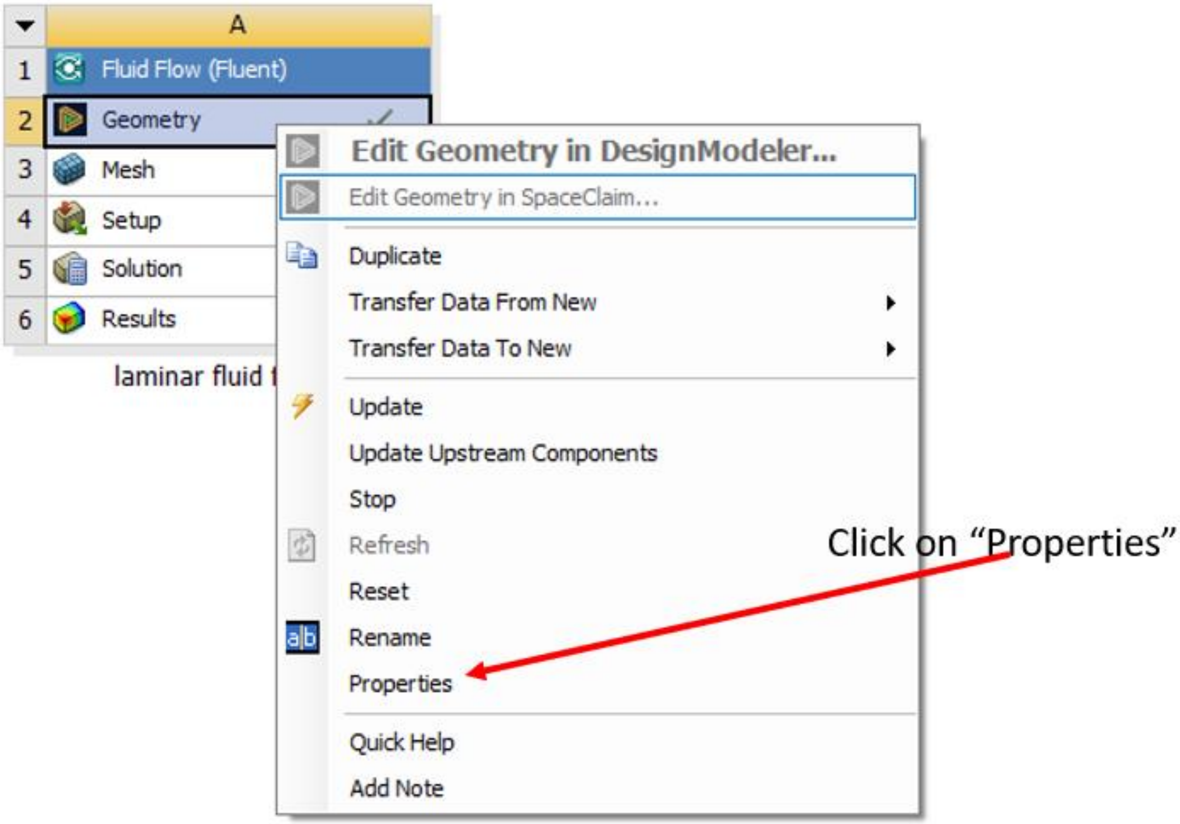
Go to “WB” SCREEN.

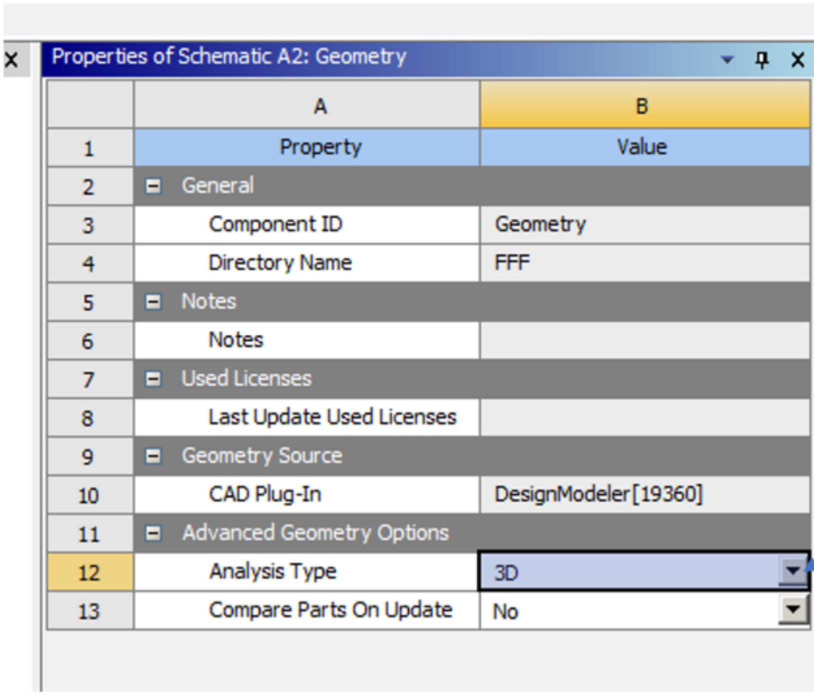


laminar fluid flow

Right click on “Geometry”

After right clicking on geometry above, the following appears.



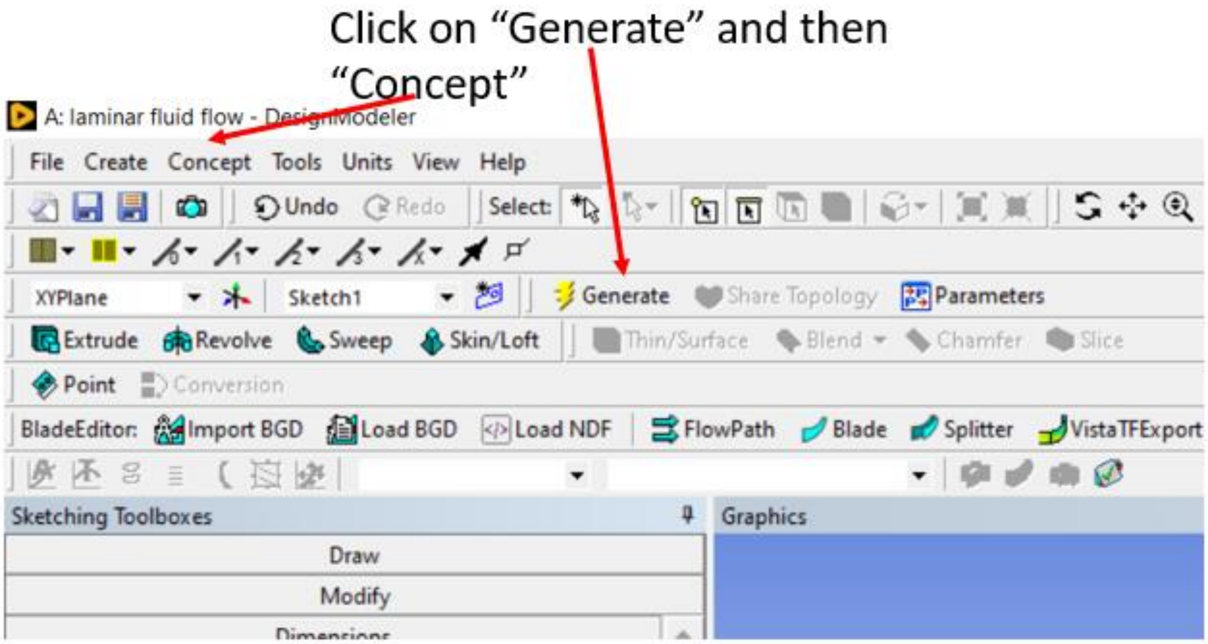


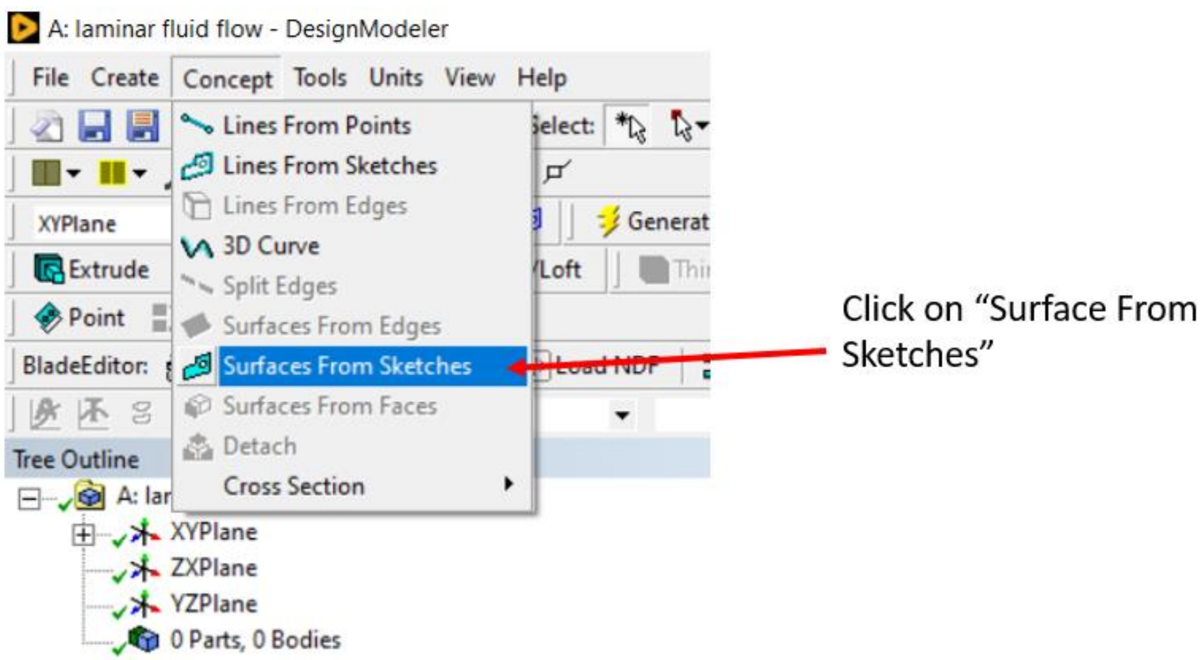
Properties of Schematic A2: Geometry

	A	B
1	Property	Value
2	General	
3	Component ID	Geometry
4	Directory Name	FFF
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Geometry Source	
10	CAD Plug-In	DesignModeler[19360]
11	Advanced Geometry Options	
12	Analysis Type	3D
13	Compare Parts On Update	No

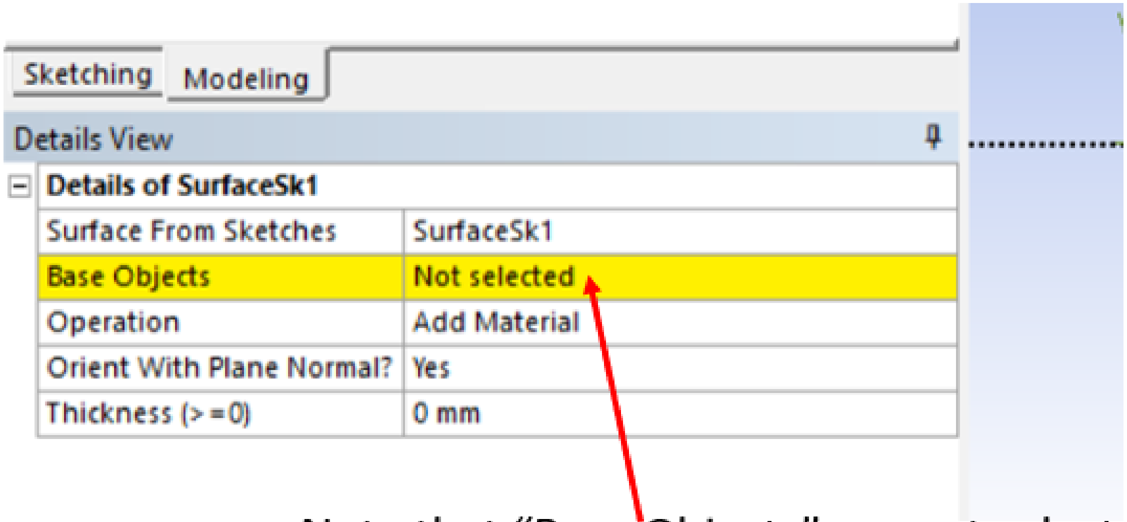
Click on this arrow and change Analysis Type to “2D”

The “2D” option was selected earlier. This section provides another example of how to do this if it was not done earlier. Basically, by using the “Properties” option above, the characteristics of the Geometry can be changed. Since “2D” was selected earlier, the “2D” option is already selected.



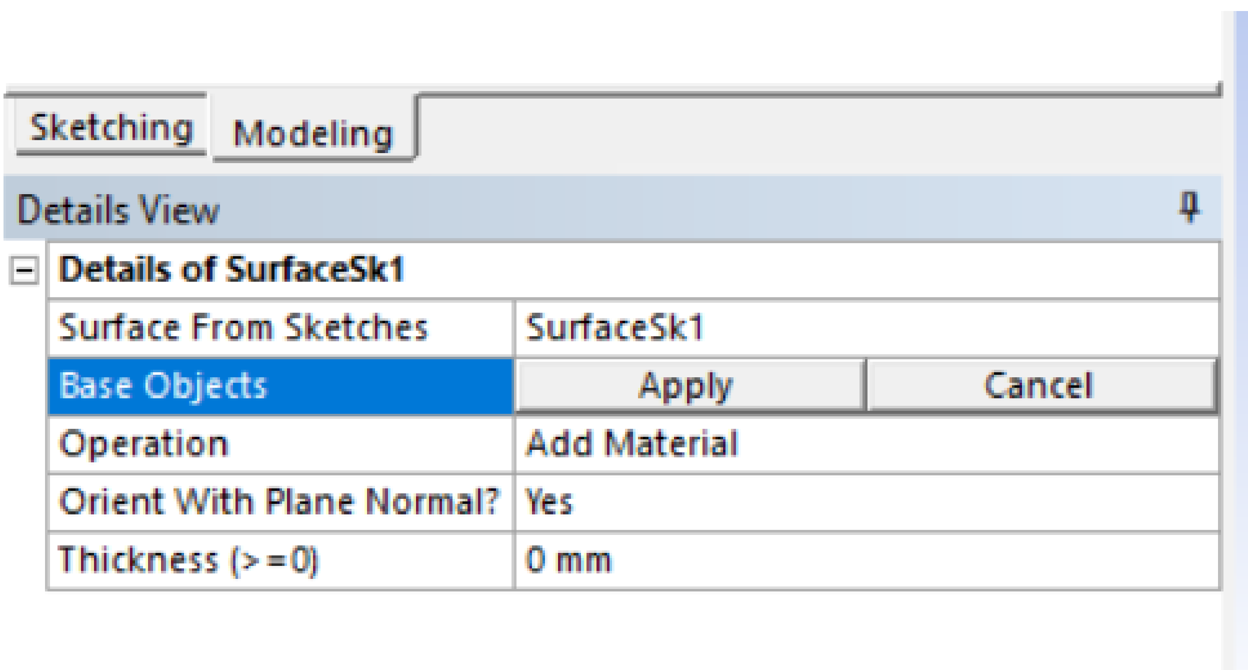


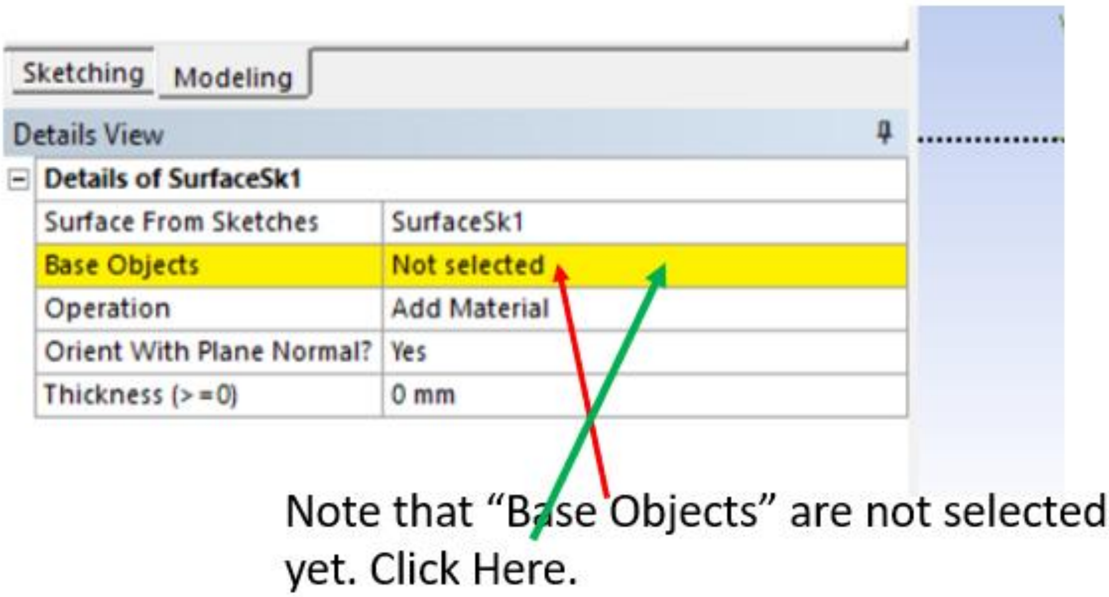
The following appears at the bottom left of screen. (The following appeared as a result of an error).



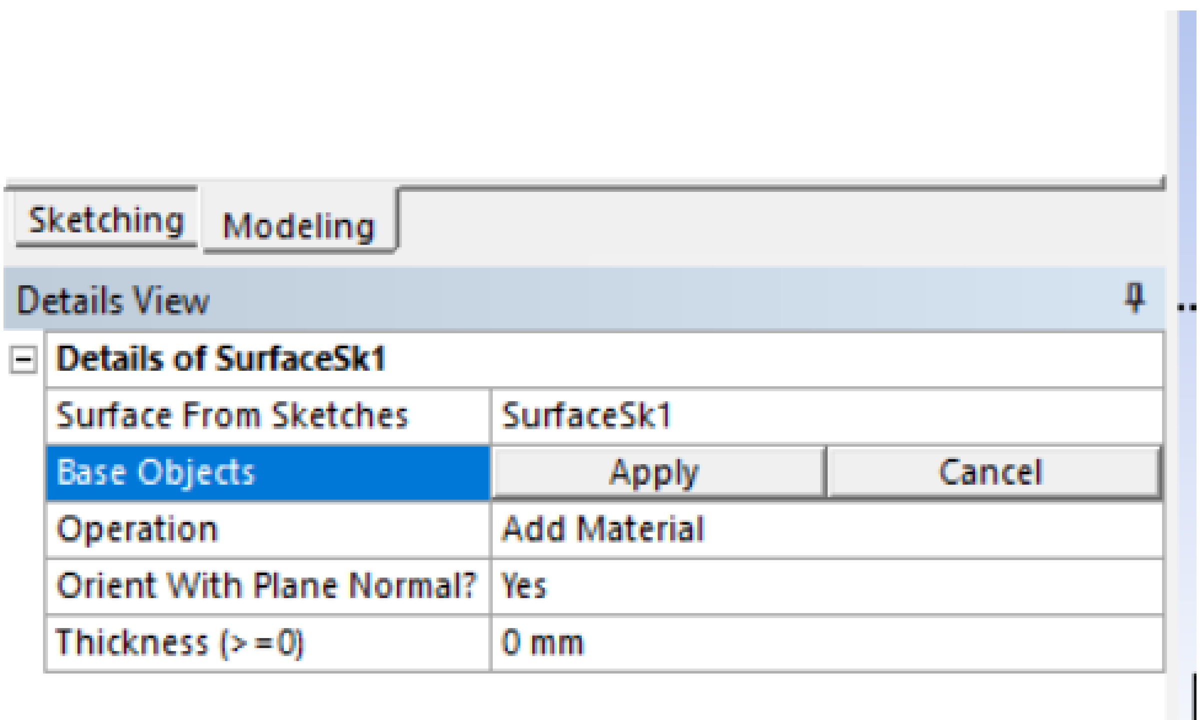
Note that "Base Objects" are not selected yet

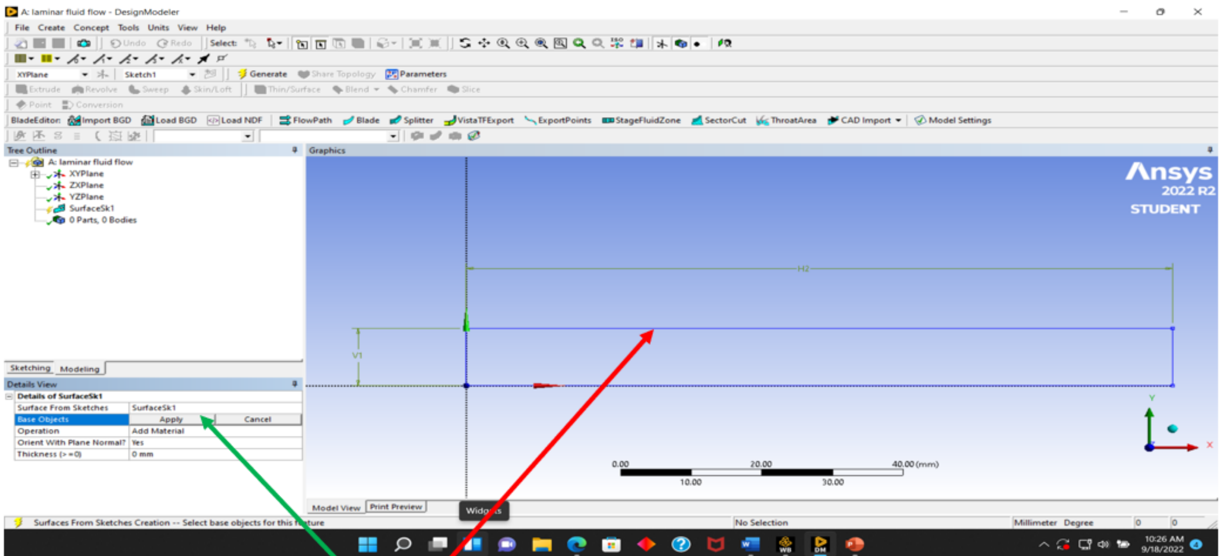
Or alternatively, the following appears on the left of screen, if no error was made.





After clicking on “Not selected” above, the screen appears as shown below.

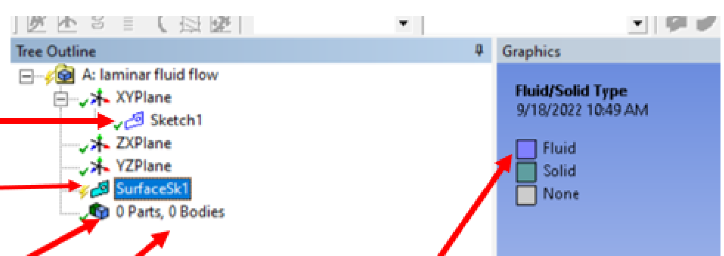




Click on this line
and then
Click on “Apply”

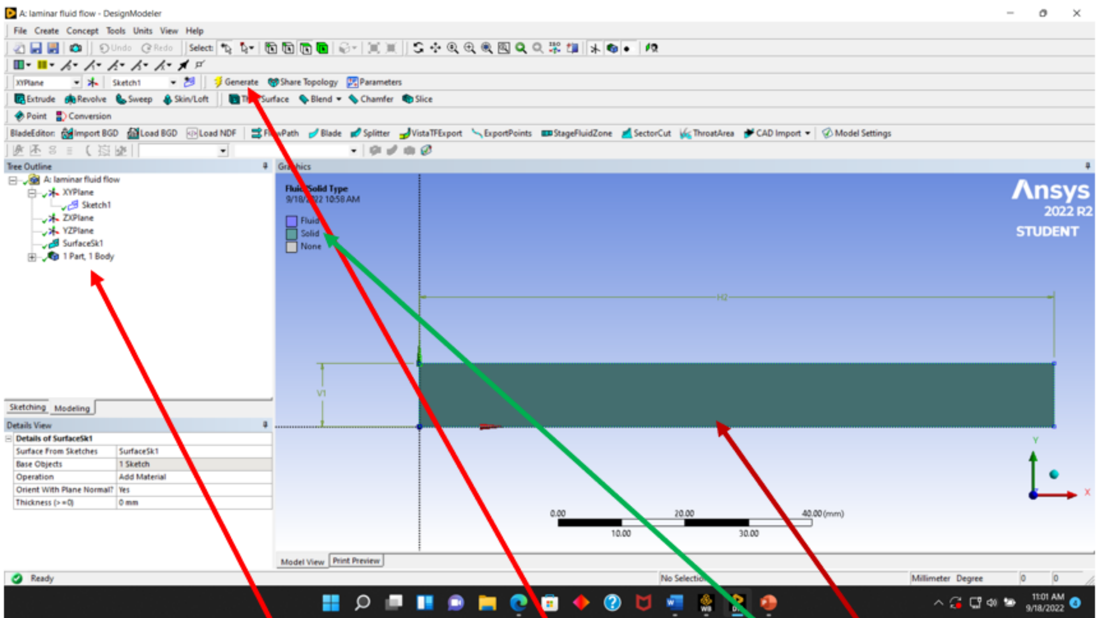
For different reasons, things may disappear. The tree on the left is used to make them appear and to keep track of them. If things disappear, click the different components of the tree on and off.

The tree diagram is showing that on XYPlane, there is Sketck1 and SurfaceSk1



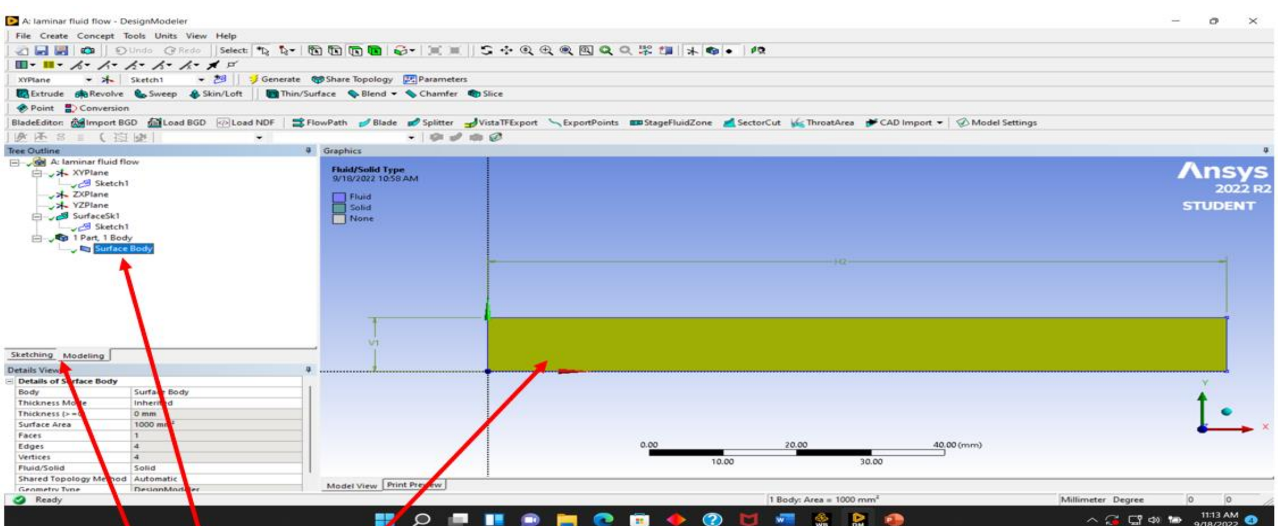
At this point, there is no parts, no Bodies

This is showing that the fluid is Blue



Click on Generate.
The surface is
converted to
1 Part, 1 body

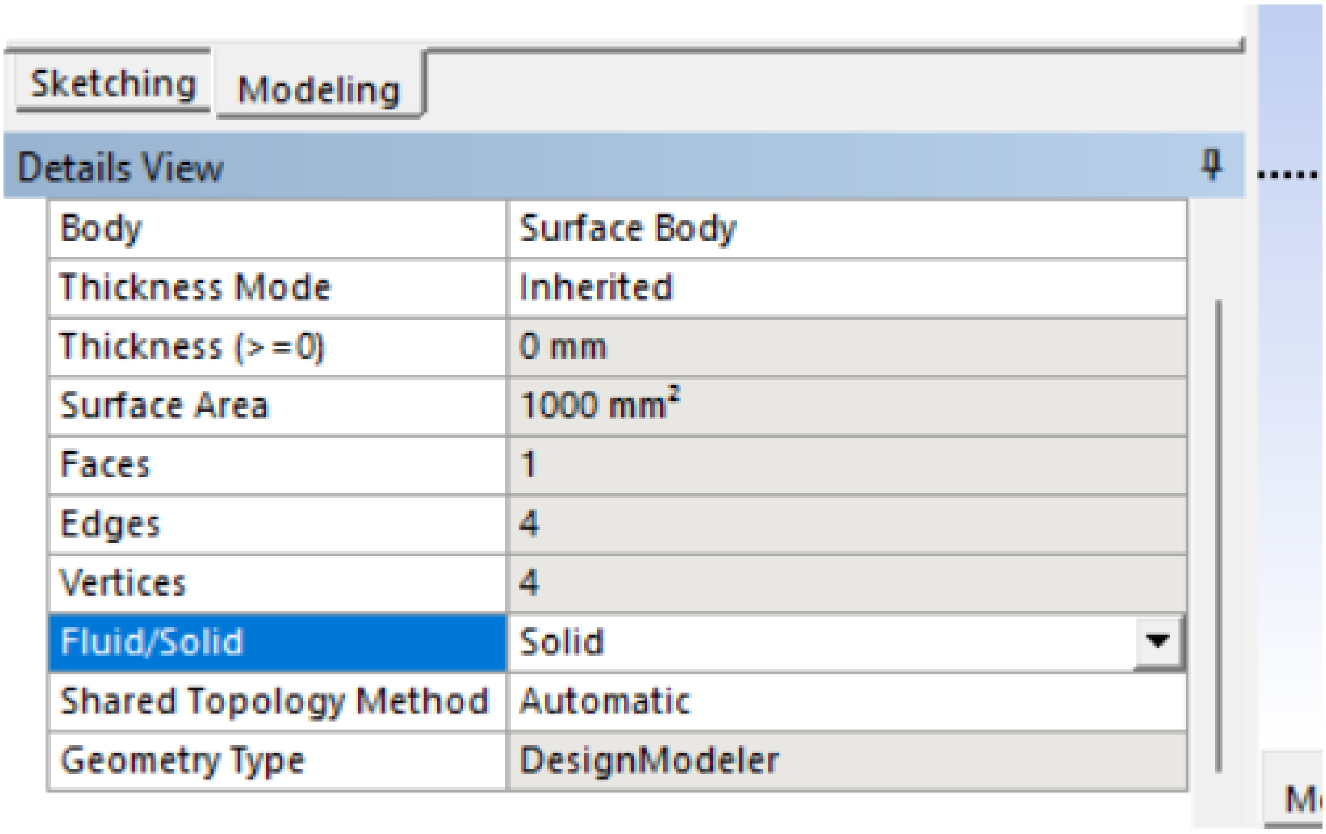
The color is changed to that of
a Solid.



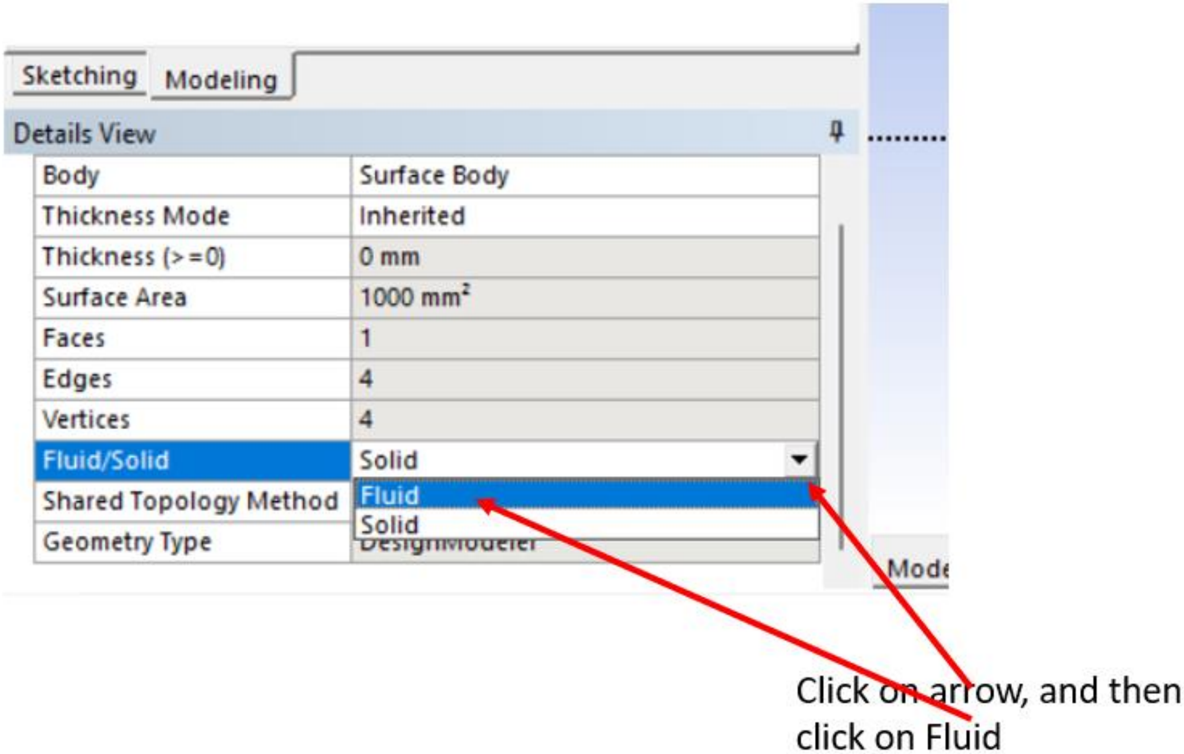
Click on "Surface Body", the
body becomes highlighted
and this box appears.

The “Surface Body” becomes visible by clicking on + next to “1 part, 1 Body”, and only then can be clicked on as defined above. In summary, whatever is clicked on tree on left of screen becomes highlighted.

At the bottom left of screen, the model information is available under “Modeling” tab as shown below.

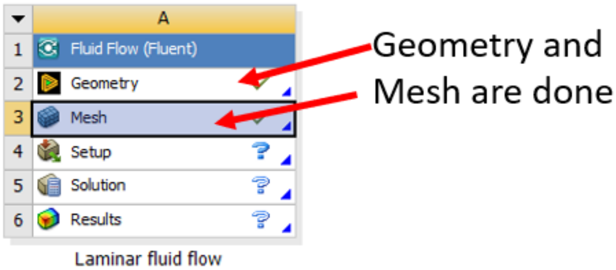


As highlighted above, in the lower left corner under modeling, the Fluid/Solid type is defined as Solid. This needs to be changed to “Fluid”.



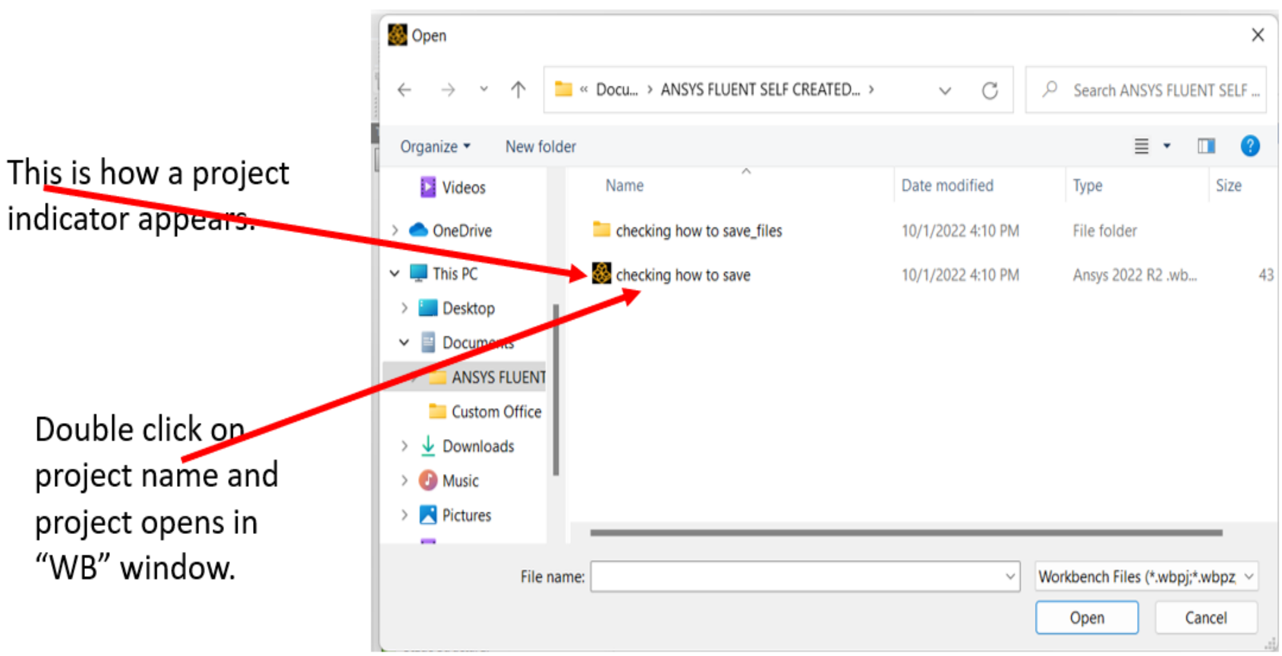
After changing to "Fluid", click on "Generate" on top to make sure the body type is changed to fluid. If this is not done, it may cause problems later in the modeling. At this point, a rectangle for fluid modeling is generated under the "Geometry" option (on the WB screen).

Since the geometry portion is now complete, minimize the "DM" window, and bring up the "WB" window. On the "WB" window, save the project under file using standard windows techniques. The work can be stopped by closing the "WB" window after saving. Closing the "WB" window closes all other windows after a time period. The work can then be recovered by starting ANSYS mechanical again and choosing the project name under file in "WB" window. The entire project will be loaded again. However, individual windows will not pop up automatically. For example, if the project after being loaded appears as shown below, it is indicated that geometry and meshing are done and are a part of the project. However, in order for the Geometry and Meshing windows to be opened, "Geometry" and "Mesh" have to be double clicked in the "WB" window as shown below.



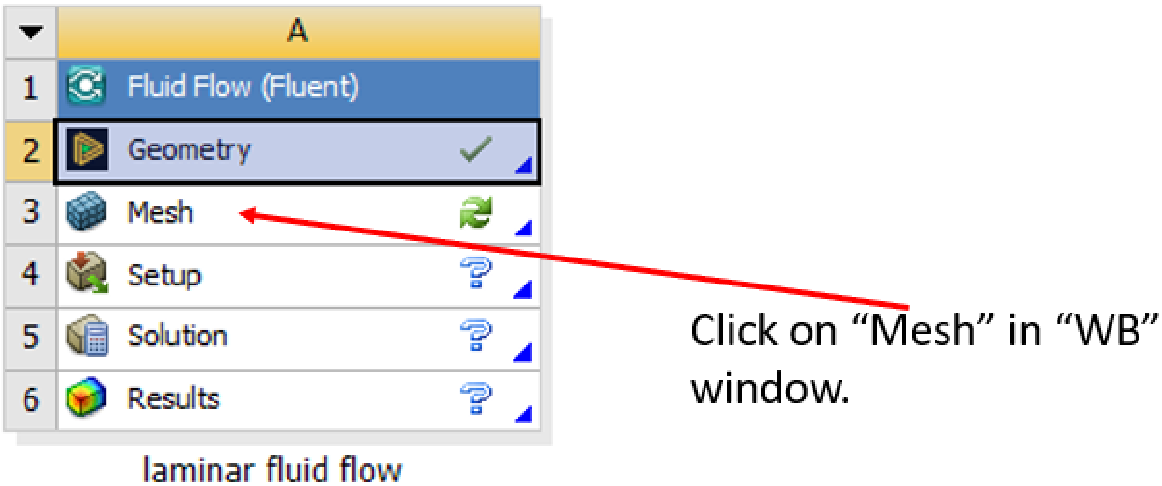
Recall that when ANSYS is started, first the WB window appears and then the DM window appears after a while. If a save job is being brought in, close the DM window, and delete the project on the WB window. After an empty WB window is available, import the data base for another job.

The window appears as shown below when an attempt is made to open the project in the "WB" window through standard windows file operations.

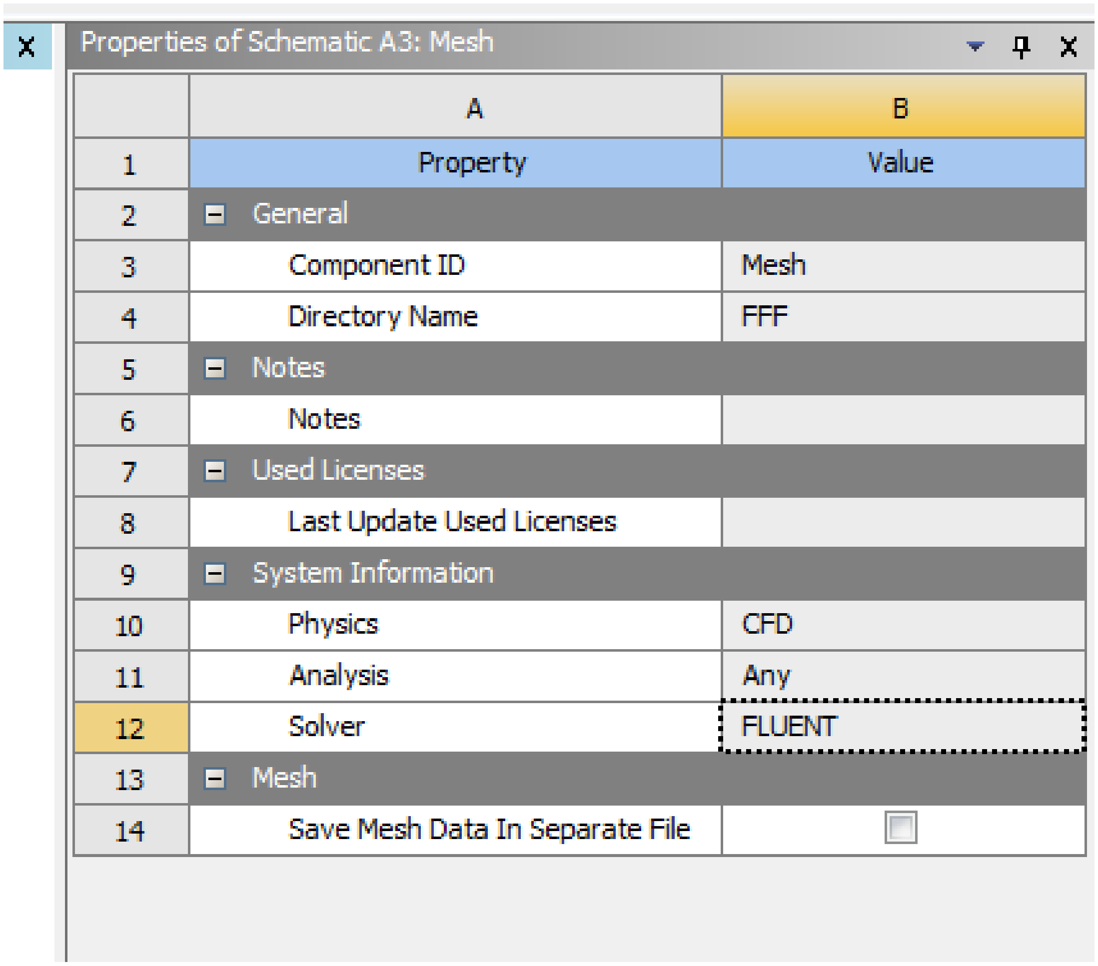


Continue creating the mesh in the next step after saving the project. Save the project at the end of each step, in order to make recovering from modeling mistakes easier.

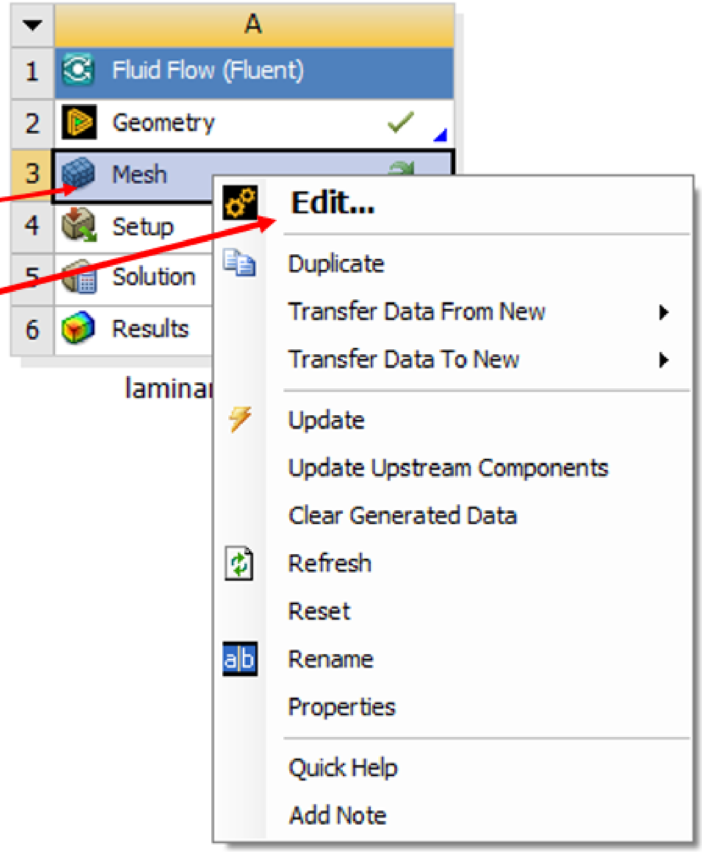
Activate the “WB” window.



The upper right side of the screen changes as shown below showing the Mesh properties.


Properties of Schematic A3: Mesh

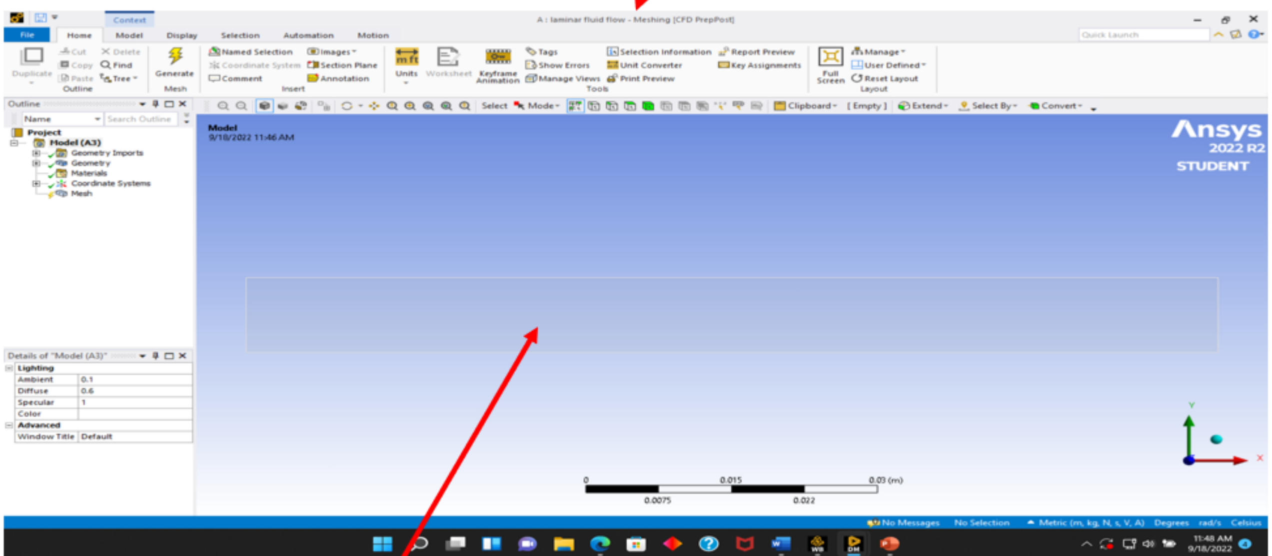
	A	B
1	Property	Value
2	General	
3	Component ID	Mesh
4	Directory Name	FFF
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	System Information	
10	Physics	CFD
11	Analysis	Any
12	Solver	FLUENT
13	Mesh	
14	Save Mesh Data In Separate File	



Click on “Mesh” and then
Right click.
The Edit box appears
Single click or perhaps double click on “Edit”, and the meshing screen appears at the bottom. This is a new screen. The appearance of this screen takes a while

The meshing screen appears as shown below.

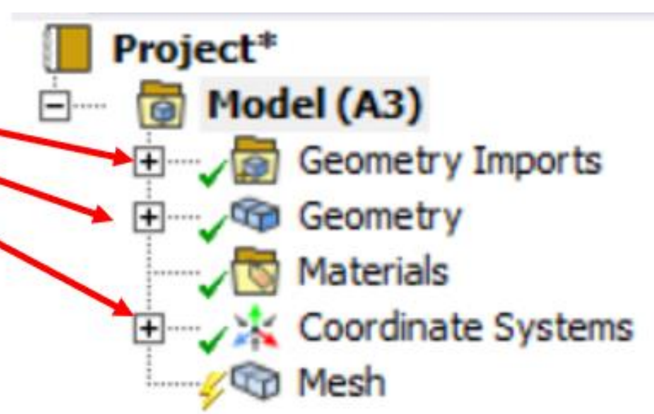
This says:
A: laminar flow – Meshing [CFD PrepPost]



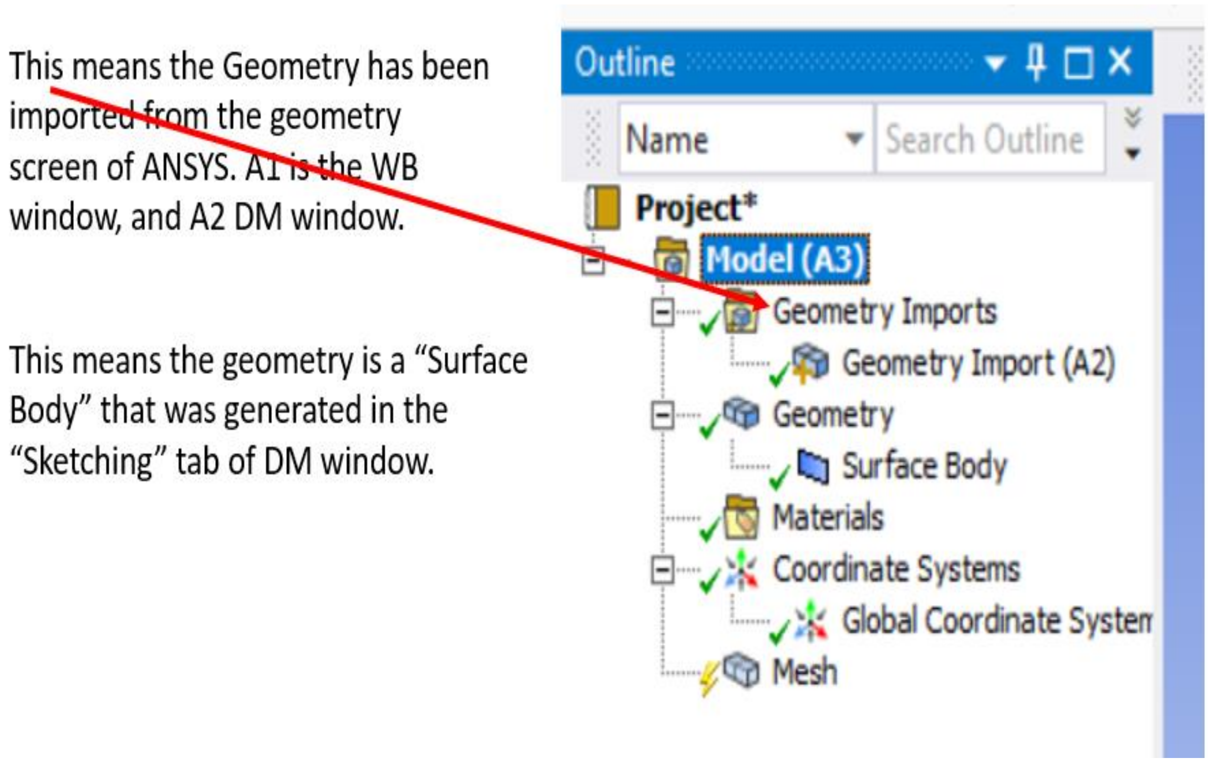
This is the surface to be meshed

The left side of screen looks as follows.

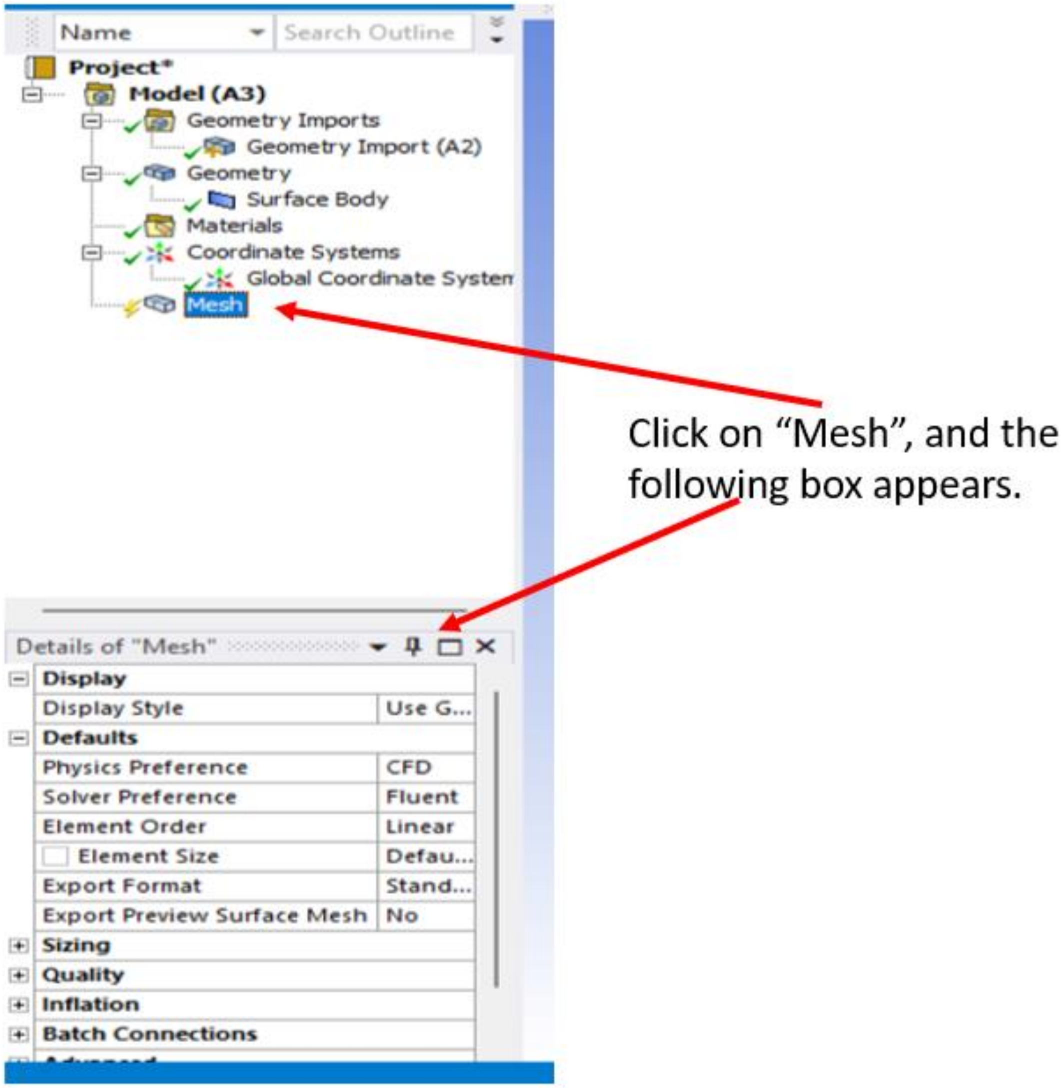
Click on all + to change it to -



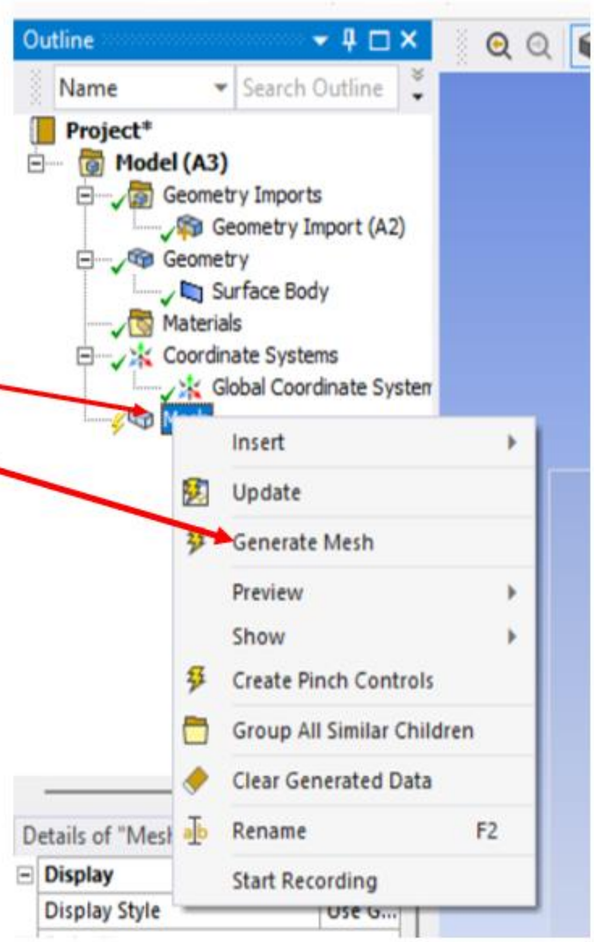
After all the + signs are changed to – signs, the screen appears as shown below.



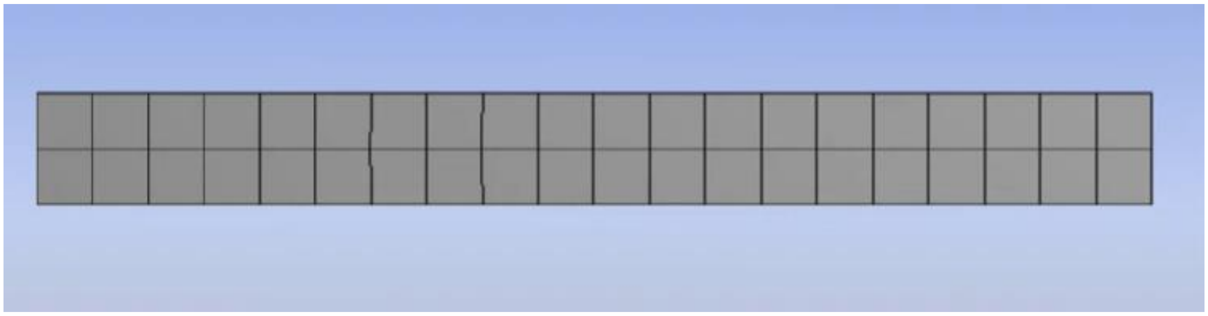
ANSYS has some default meshing features. The default meshing features will be used first.



Right Click on “Mesh” and
then
Click on “Generate Mesh”

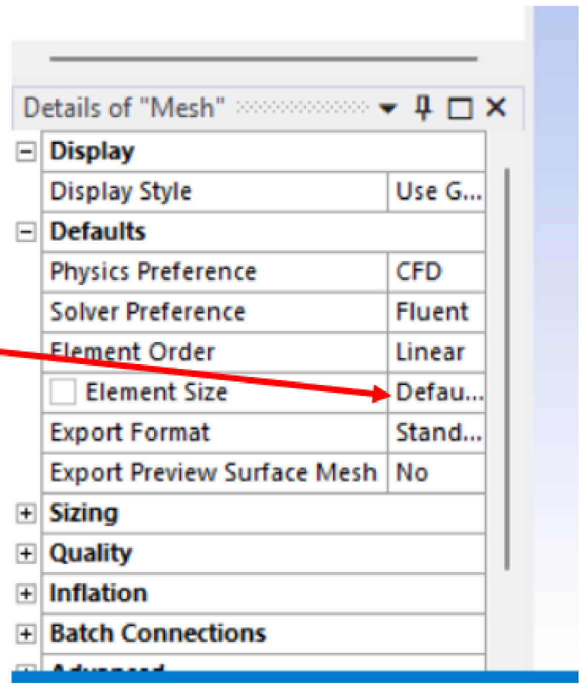


The following default mesh is generated.

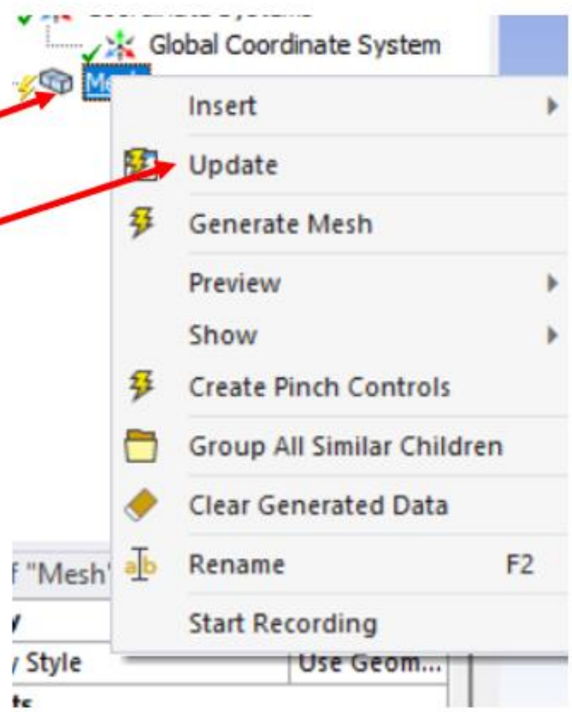


The following is on the lower left corner of the mesh screen.

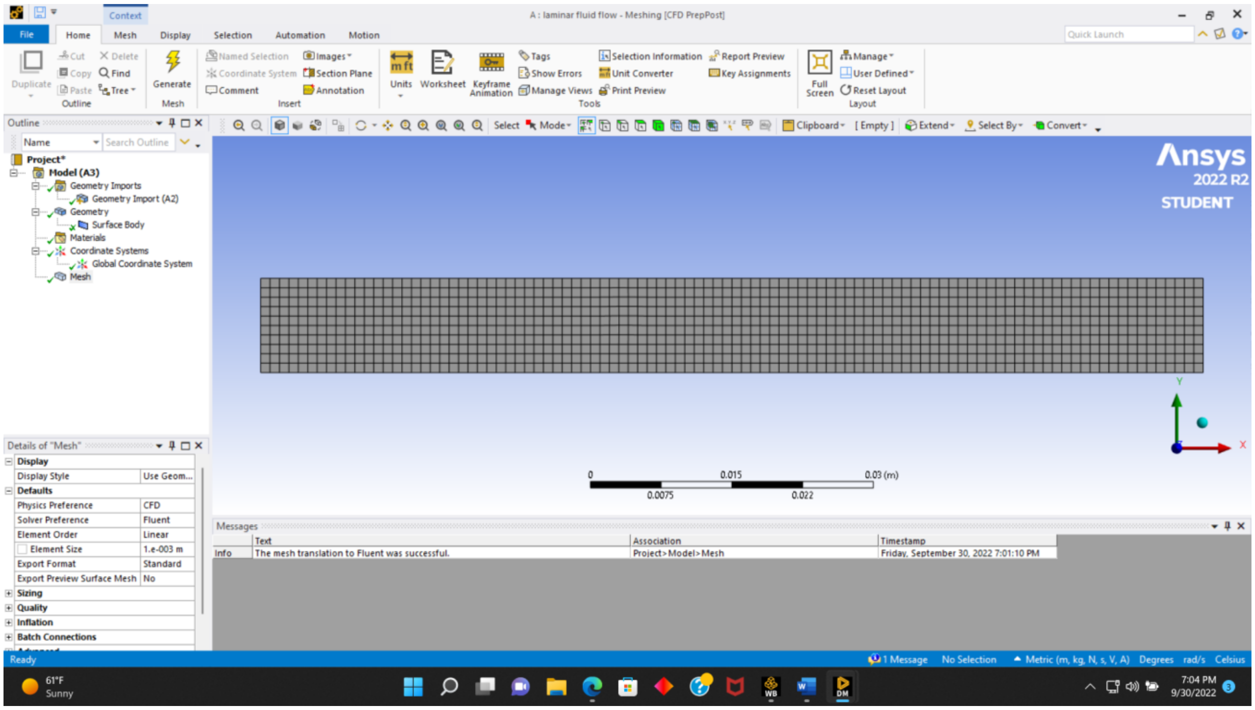
Click here and change the mesh size to 0.001



Right click on “Mesh” and then click on “Update”



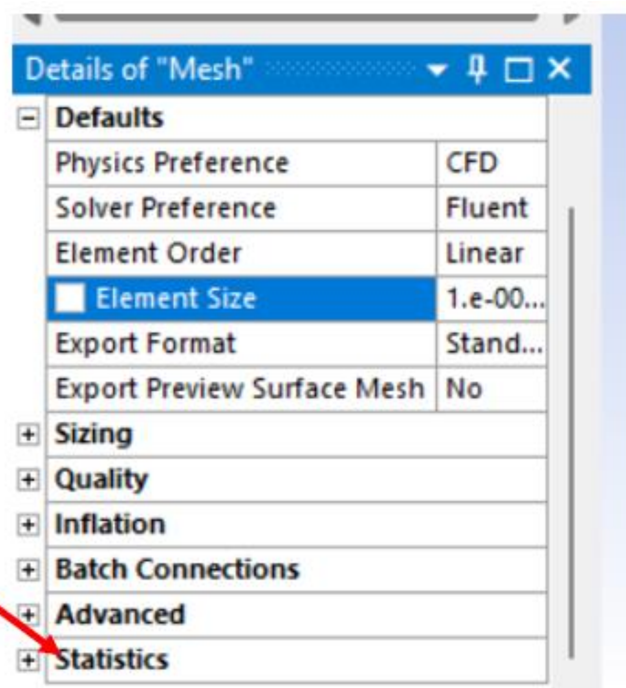
The mesh is updated as shown below.



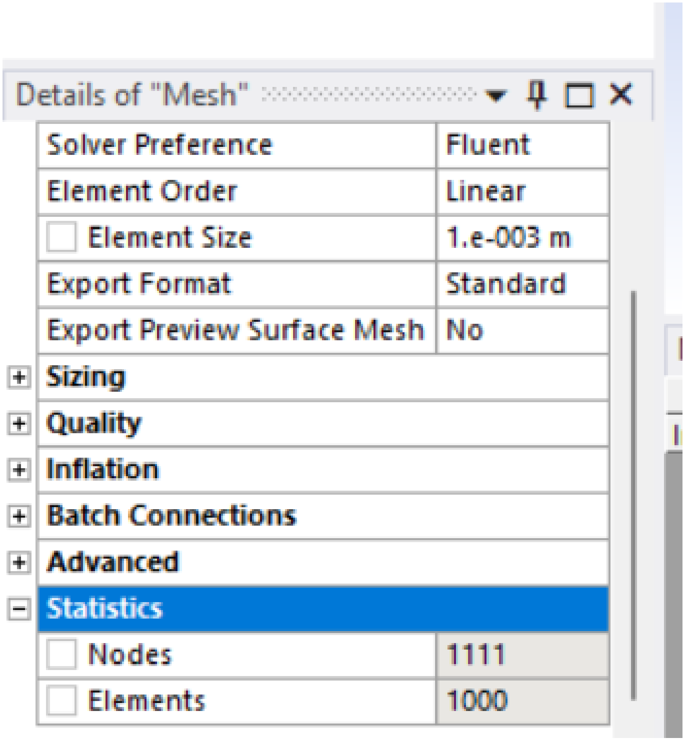
The correctness of the mesh for the application must be checked.

At the bottom of the table on the lower left corner, the mesh statistics can be checked.

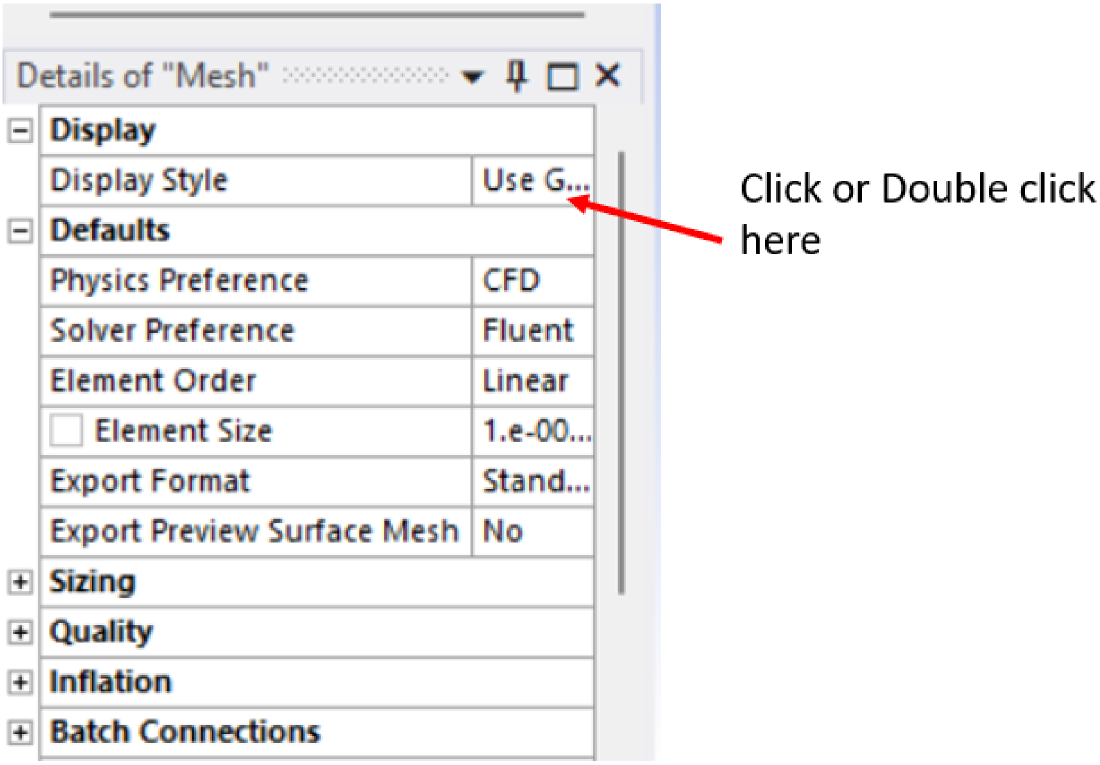
Double Click on "Statistics



After clicking on the Statistics, the following appears. It is shown that the number of nodes is 1111, and the number of elements is 1000.

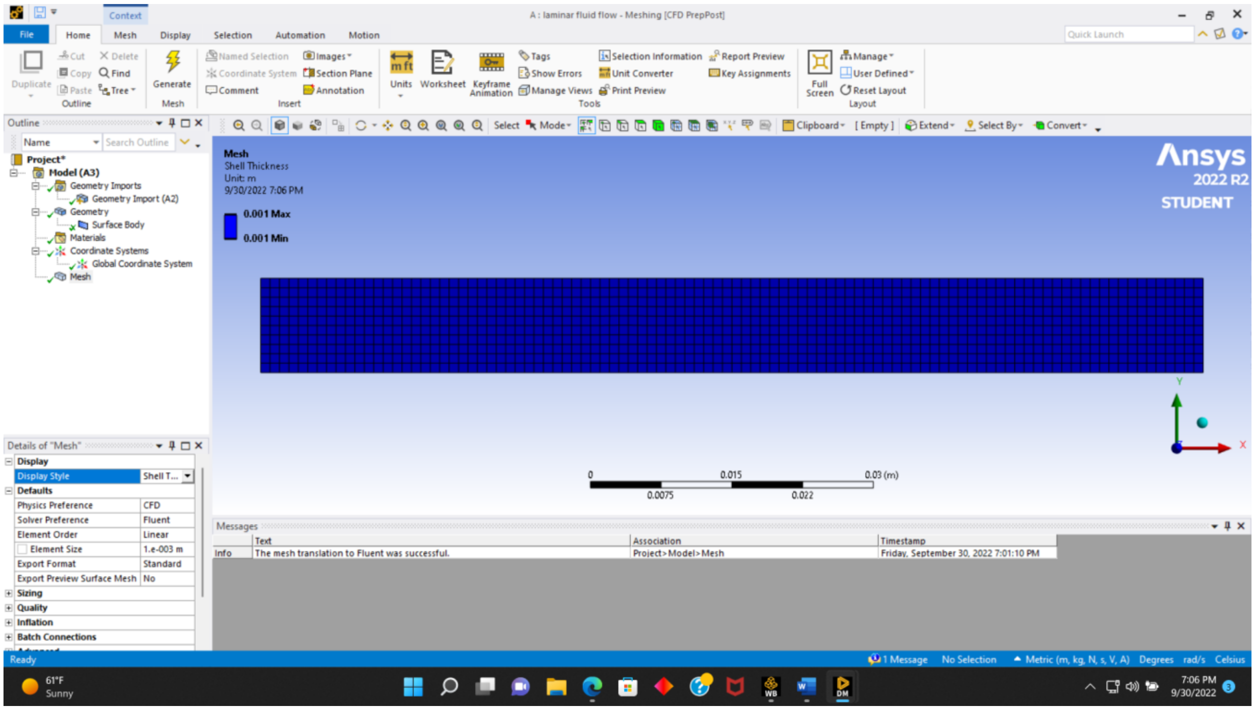


Do the following in the "Details of "Mesh" at the bottom left-hand side of the screen.



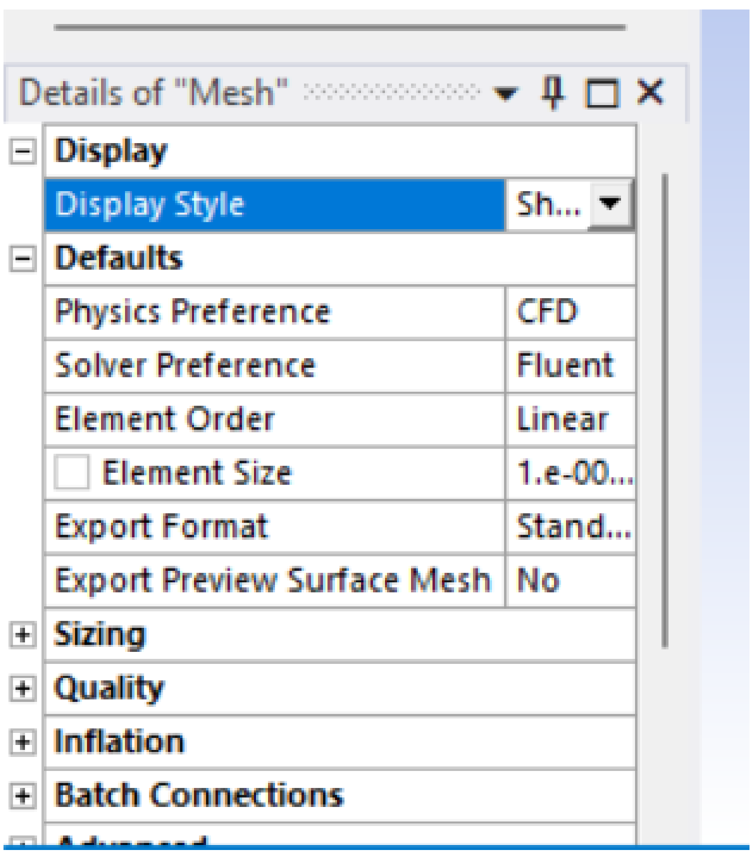
"Use G…" above, stands for "Use Geometry".

The following appears.

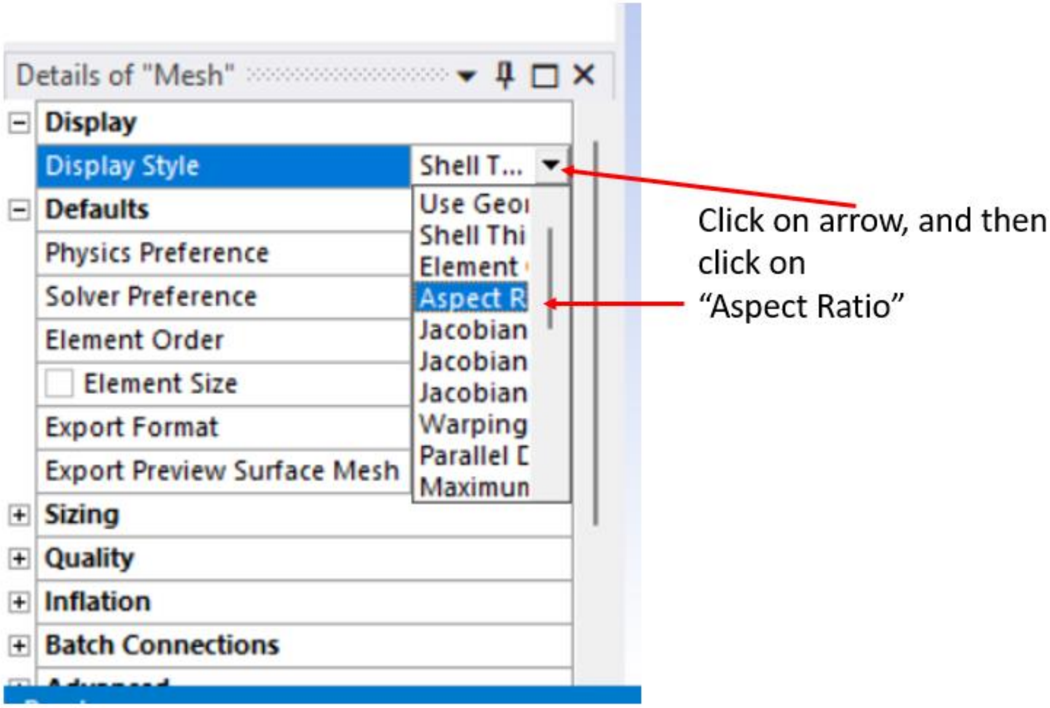


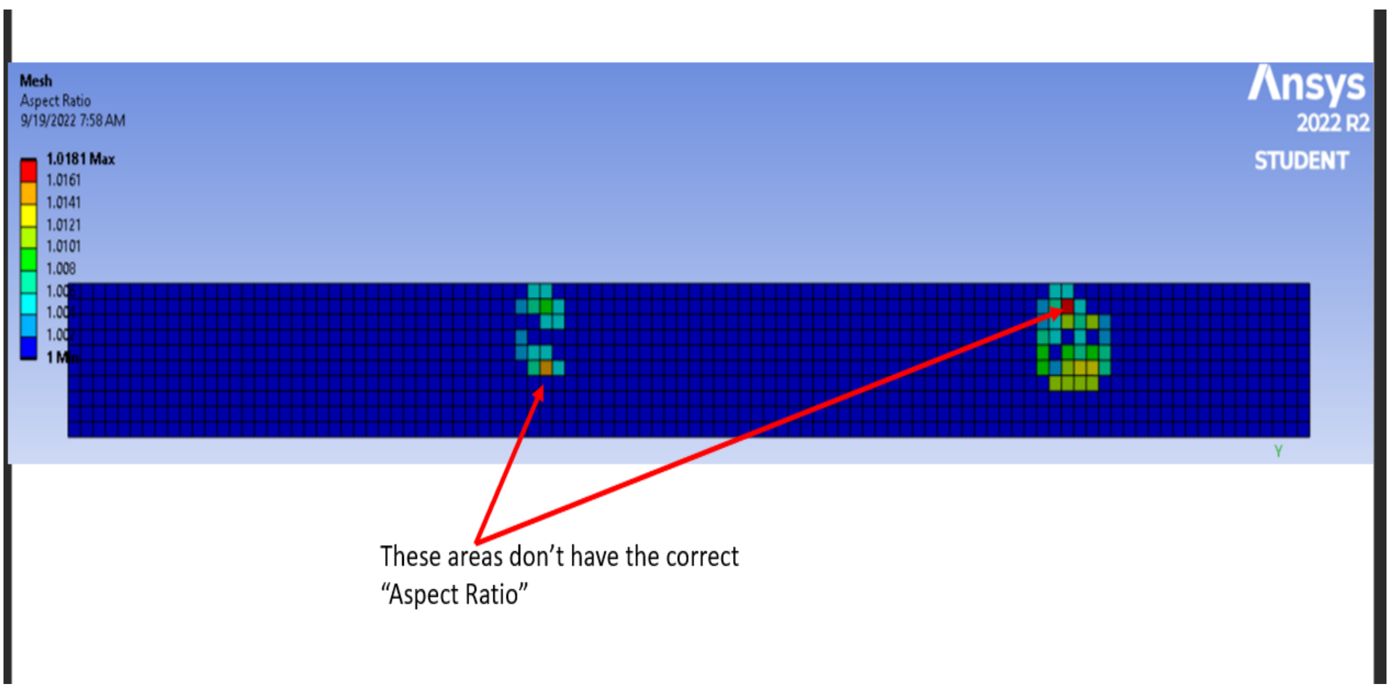
The following 2 screen shots are the zoomed in sections of the above.

The meshing is now uniform.

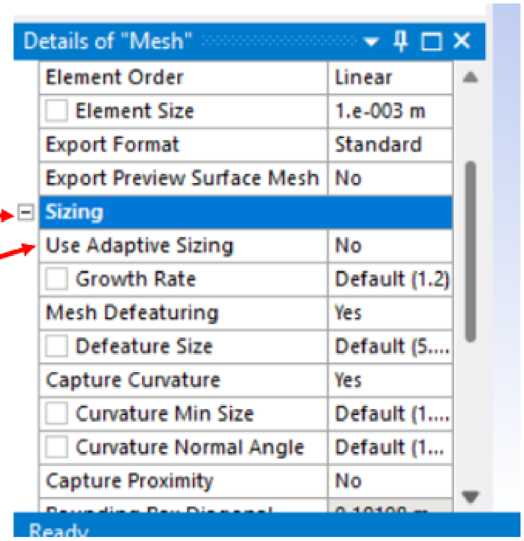


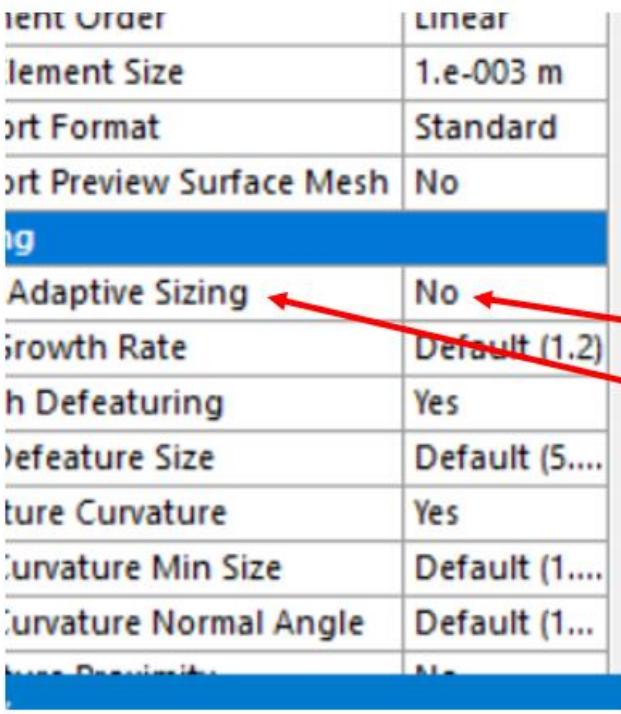
In order to check the meshing "Aspect Ratio" do the following.





Click on the + size here to turn it into a – sign as show. When it is negative, The Sizing options appear. The sizing option of interest in this example is "Adaptive Sizing"

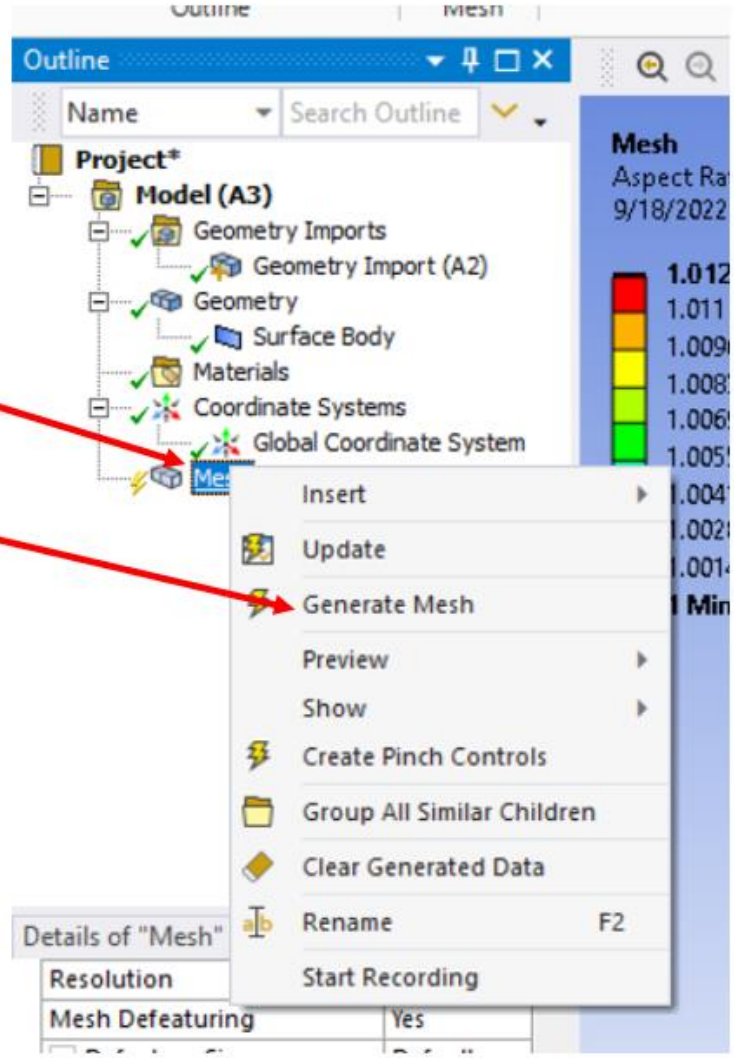




Click on "No" next to "Adaptive Sizing" to turn it into "Yes"

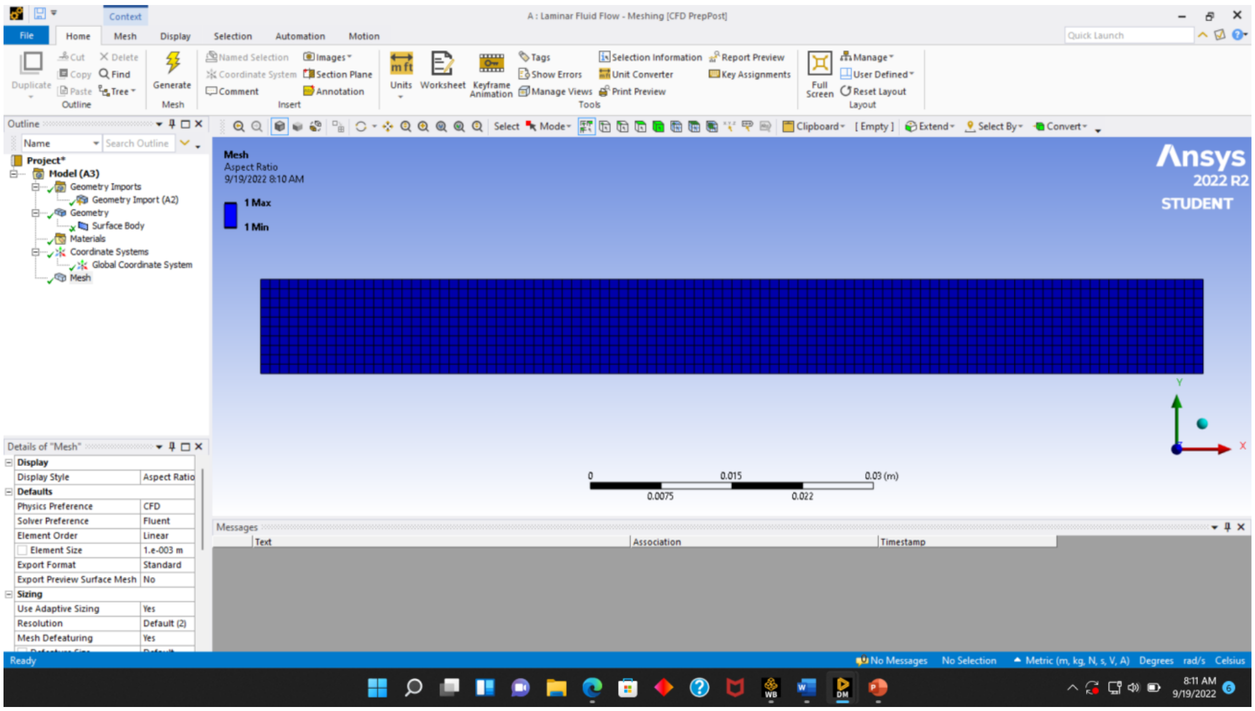
After turning "Adaptive Sizing" on by changing the "No" to "Yes", do the following.

Right click on
“Mesh”
And then double
click on
“Generate Mesh”



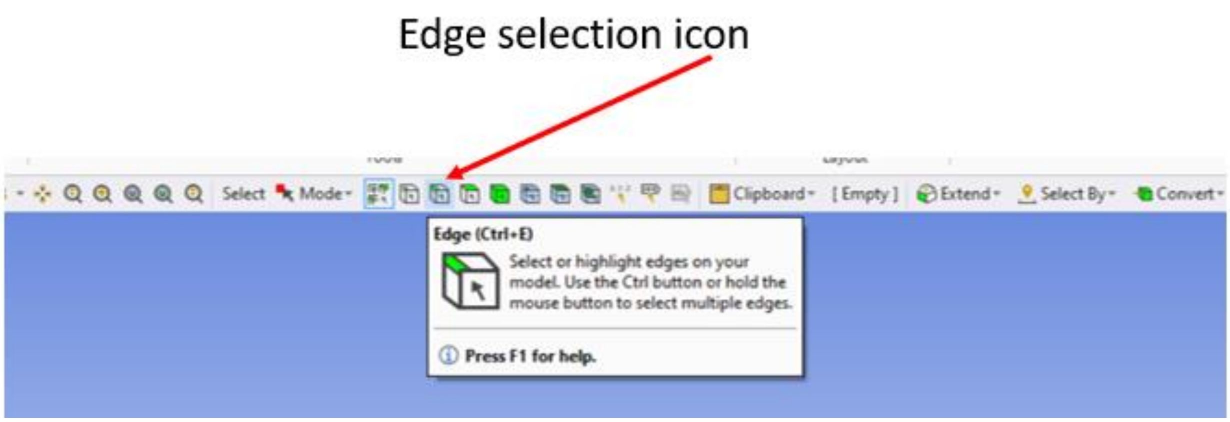
The following mesh is generated. The mesh is now better, because the minimum and maximum size is now 1.

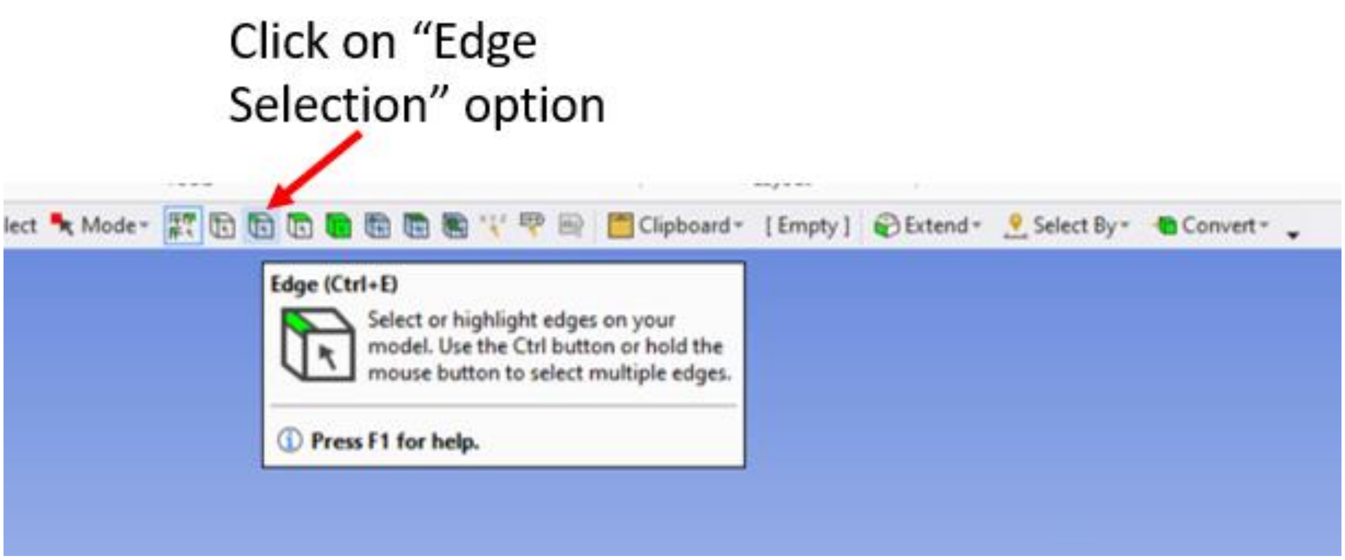
The better mesh would have been 1 by 1. The example model is 1 by 1 and is as shown below.


A : Laminar Fluid Flow - Meshing [CFD PrepPost]
Mesh
Aspect Ratio
9/19/2022 8:10 AM
1 Max
1 Min
Ansys
2022 R2
STUDENT
0
0.015
0.03 (m)
0.0075
0.022
Details of "Mesh"
Display
Display Style
Aspect Ratio
Defaults
Physics Preference
CFD
Solver Preference
Fluent
Element Order
Linear
Element Size
1.e-003 m
Export Format
Standard
Export Preview Surface Mesh
No
Sizing
Use Adaptive Sizing
Yes
Resolution
Default (2)
Mesh Defeaturing
Yes
Project*
Model (A3)
Geometry Imports
Geometry Import (A2)
Geometry
Surface Body
Materials
Coordinate Systems
Global Coordinate System
Mesh
Messages
Text
Association
Timestamp

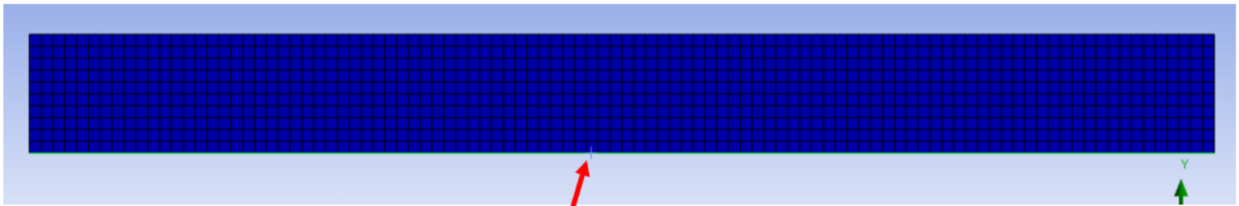
In the Mesh window, the boundary tools are shown below. The boundary tools are used to select the model boundary.

In this example, the edge select is used.



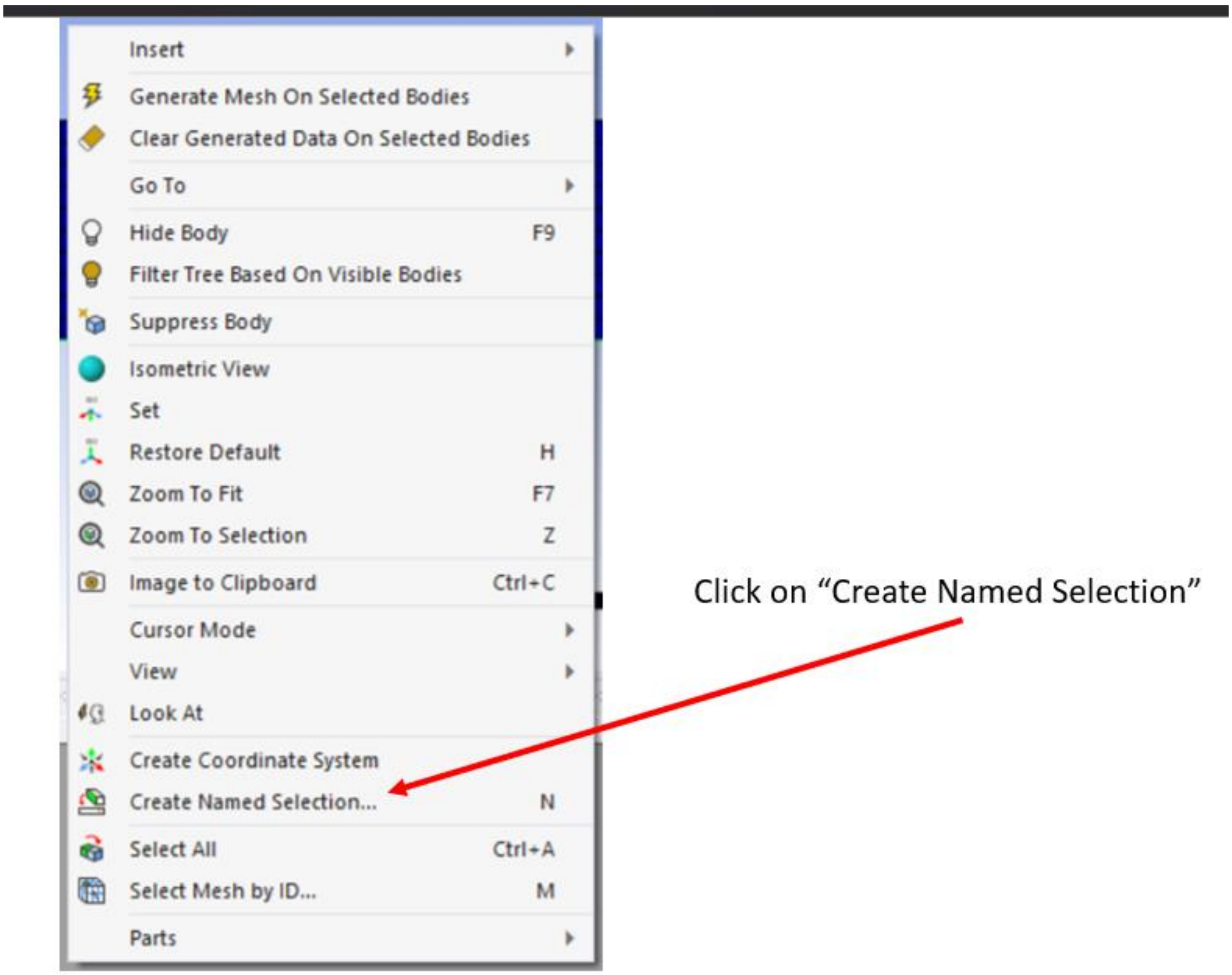


After selecting the edge above, the pointer changes. Click on the bottom of the rectangle.



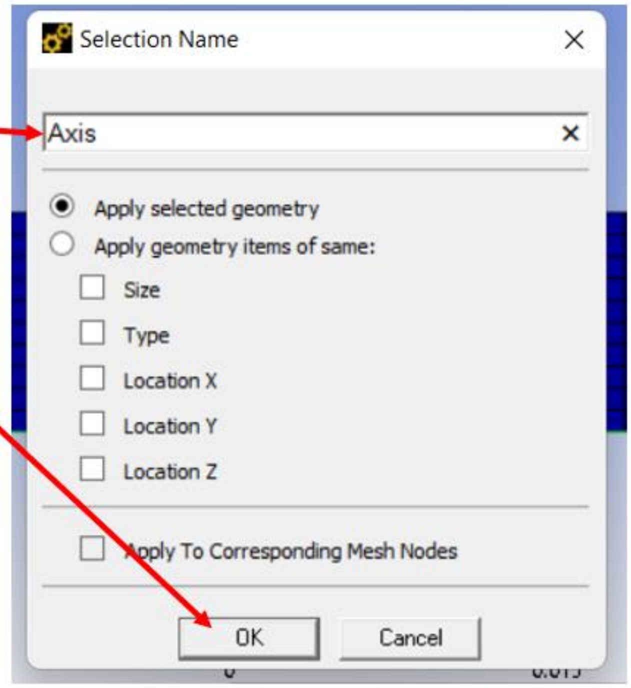
After selecting the edge option above,
click on the lower edge of rectangle.
The lower edge becomes green which
is not quite visible, but is dimly visible

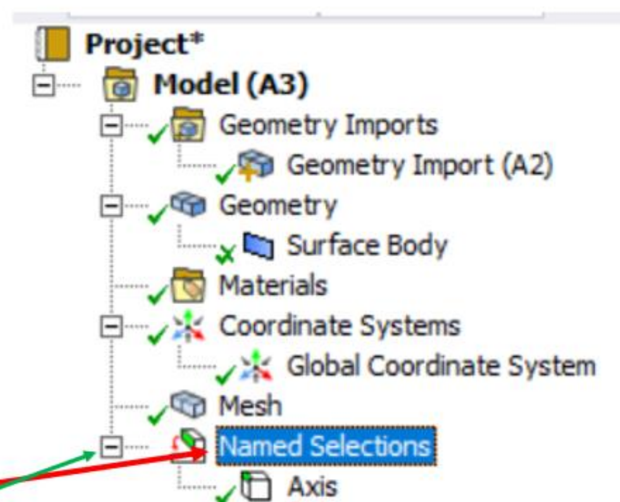
After selecting the lower edge (after which the lower edge becomes green highlighted), right click and the following appears.



After clicking on “Create Named Selection”, the following box appears, and the lower edge is given the name “Axis”.

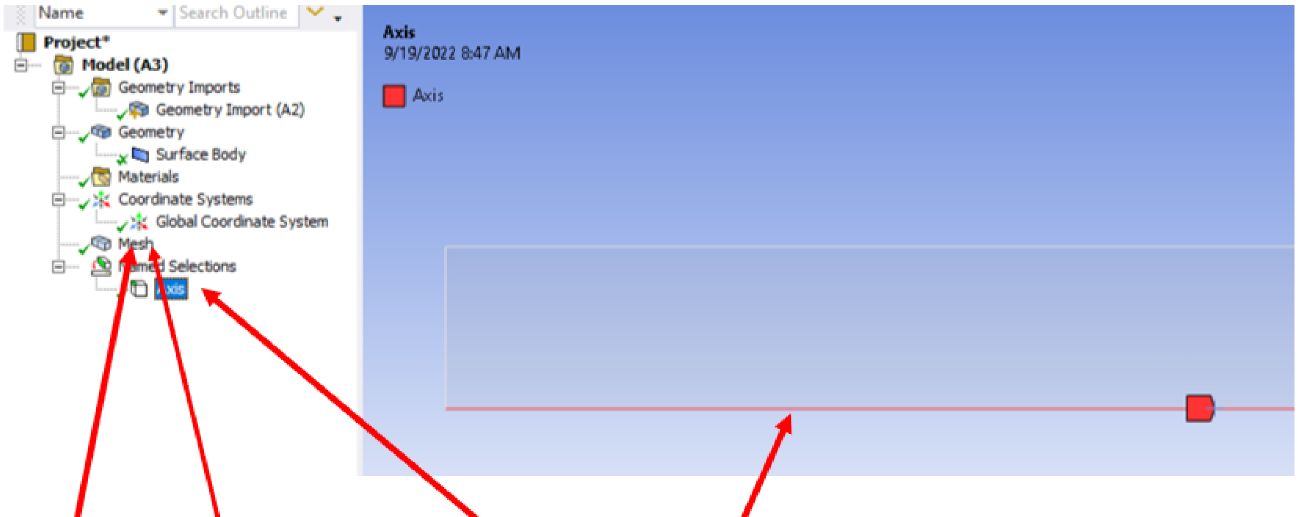
Input the name “Axis”, and then click on “OK”





“Named selections” is added.
Click on + to turn it into –
The newly named edge “Axis” becomes visible

The upper left of the meshing screen contains a tree. By clicking on various parts of the tree, things can be made to appear and disappear in order to be able to see various aspects of the model. For example, the following makes the mesh to disappear and the lower edge visible.

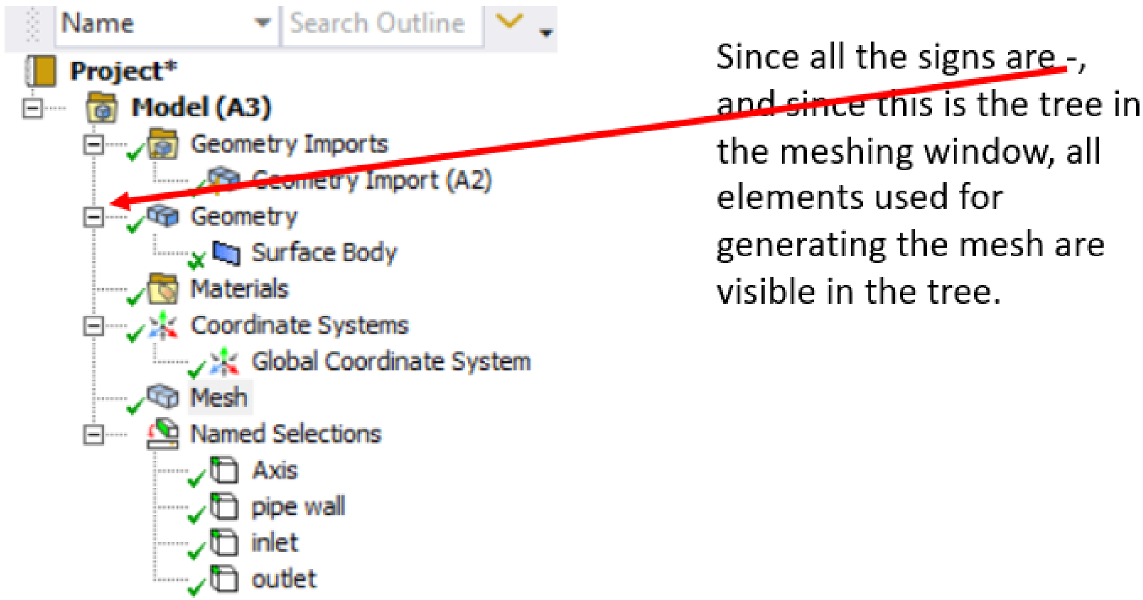


Mesh is de-highlighted. As a result mesh becomes invisible. To make the mesh visible again, click on "Mesh"

Axis is highlighted. As a result, Axis is visible.

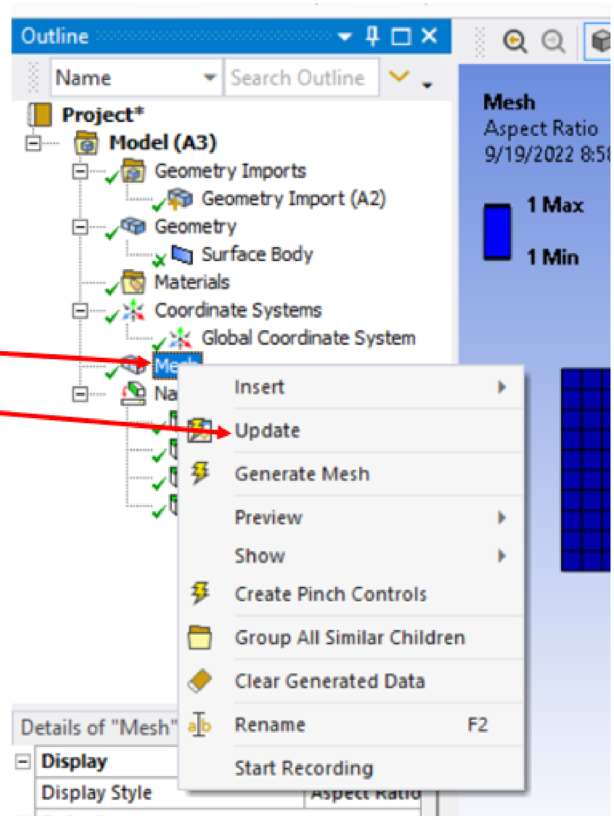
Similarly, name the top of the rectangle as "pipe wall", the left edge as "inlet" and the right edge as "outlet".

The tree diagram appears as shown below after the creation of the named edges.



Check all the named boundaries by making them visible one at a time through the use of the tree. Make sure there are no duplications in the names. If there are duplications, there will be trouble later.

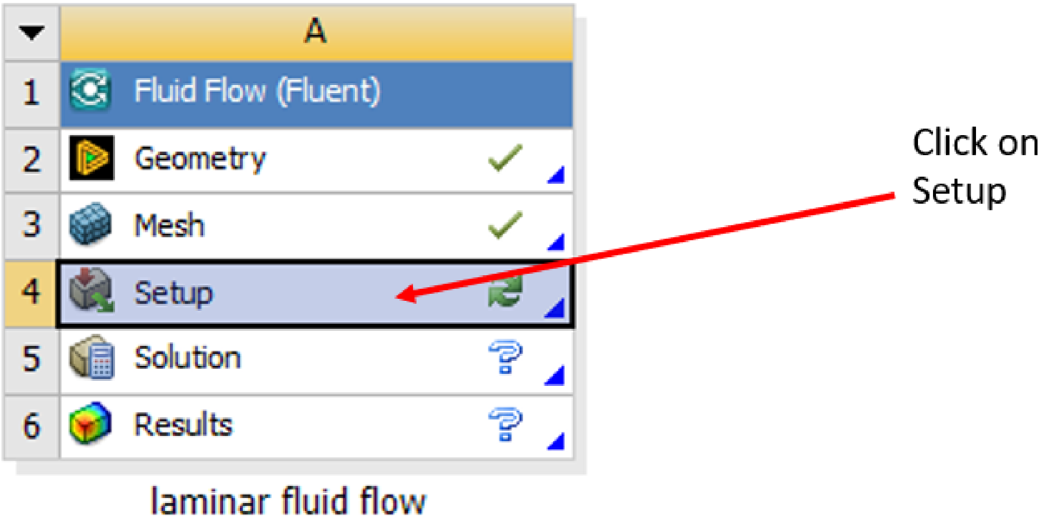
Right click on "Mesh" and click on "Update". The program will generate a message that the translation of mesh to fluent was successful



Always look for the above message to make sure the meshing was successful, and if the message does not appear or says meshing is unsuccessful, fix the problem or the model will crash.

Minimized the mesh window. Save the project in "WB" window.

Click on 'Setup" in "WB" window as shown below in order to bring up a "f" window. "f" stands for fluid and that is where the characteristics of the fluid model will be defined.

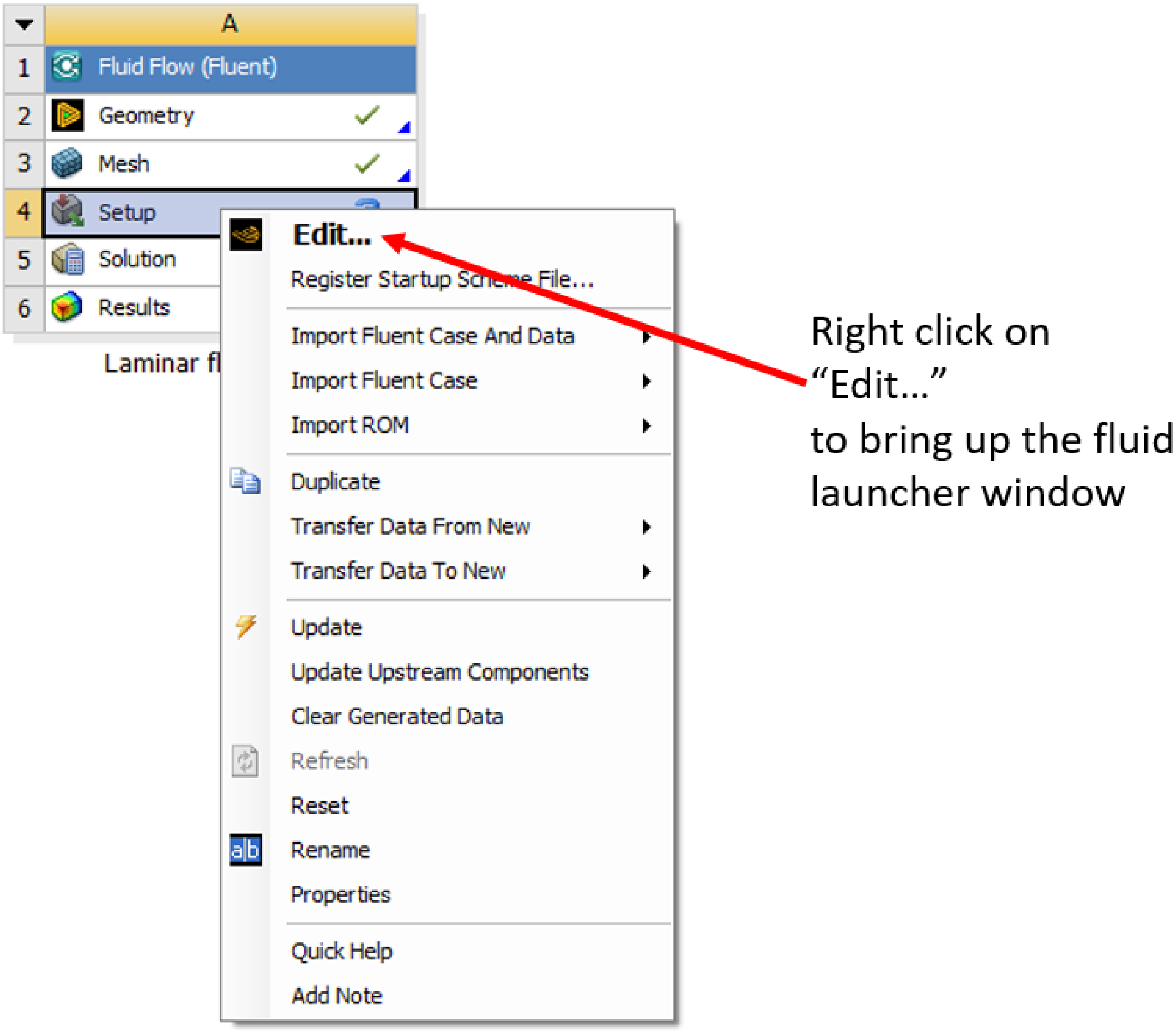


After clicking on "Setup" in the "WB" window, the following appears on the upper right-hand side of the "WB" window.

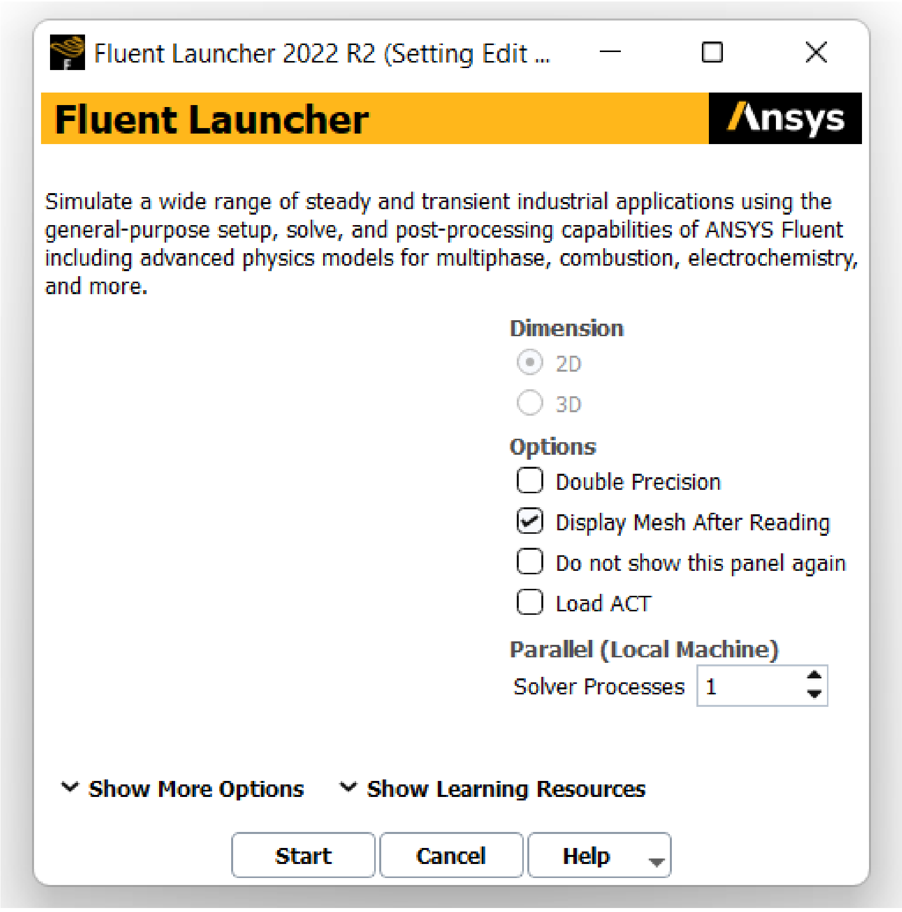
Properties of Schematic A4: Setup

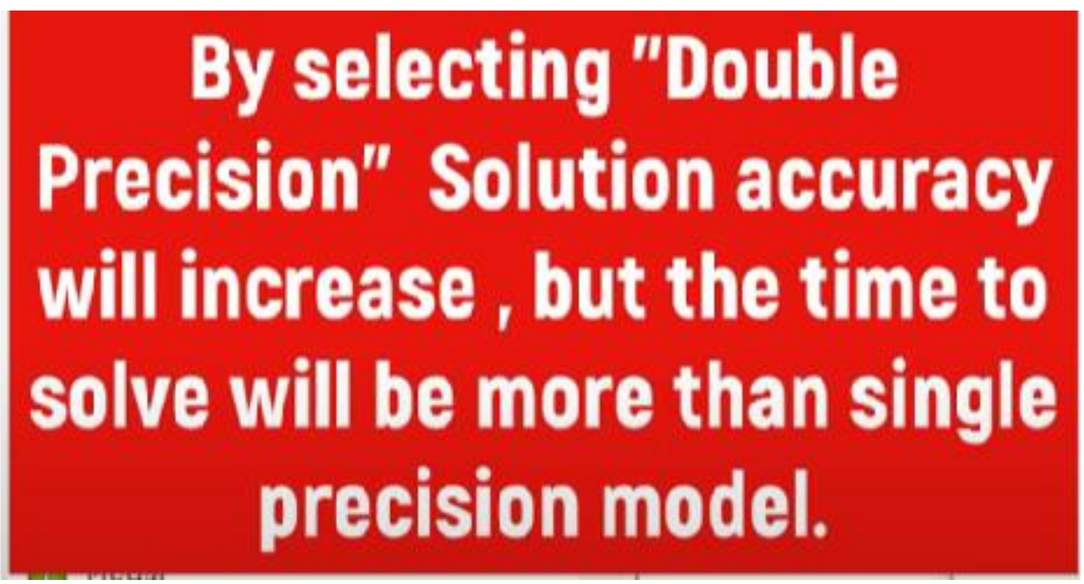
	A	B
1	Property	Value
2	⊟ General	
3	Component ID	Setup
4	Directory Name	FFF
5	Precision	Single Precision
6	Show Launcher at Startup	☑
7	Display Mesh After Reading	☑
8	Embed Graphics Windows	☑
9	Use Workbench Color Scheme	☑
10	Load ACT Start Page	☐
11	Environment Path	
12	Setup Compilation Environment for UDF	☑
13	Use Job Scheduler	☐
14	Run Parallel Version	☐
15	UDF Compilation Script Path	$(FLUENT_ROOT)\$(ARCH)\udf.bat
16	⊟ Notes	
17	Notes	
18	⊟ Used Licenses	
19	Last Update Used Licenses	
20	⊟ Others	
21	Generate Output Case File	☑

Right click on “Setup”, and then right click on “Edit…” and a fluid launcher window appears.

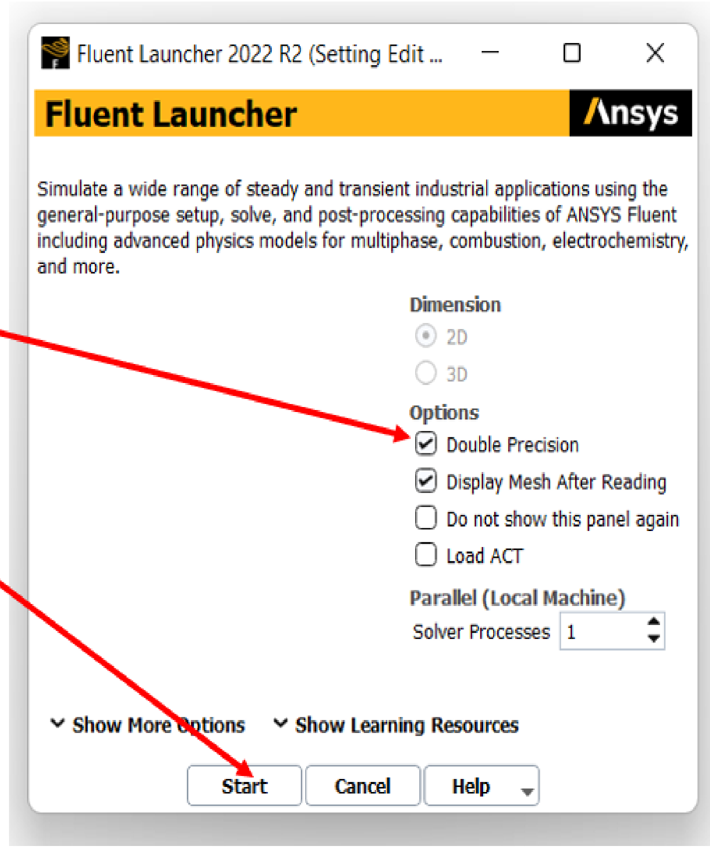


The following is the "Fluid Launcher" window.

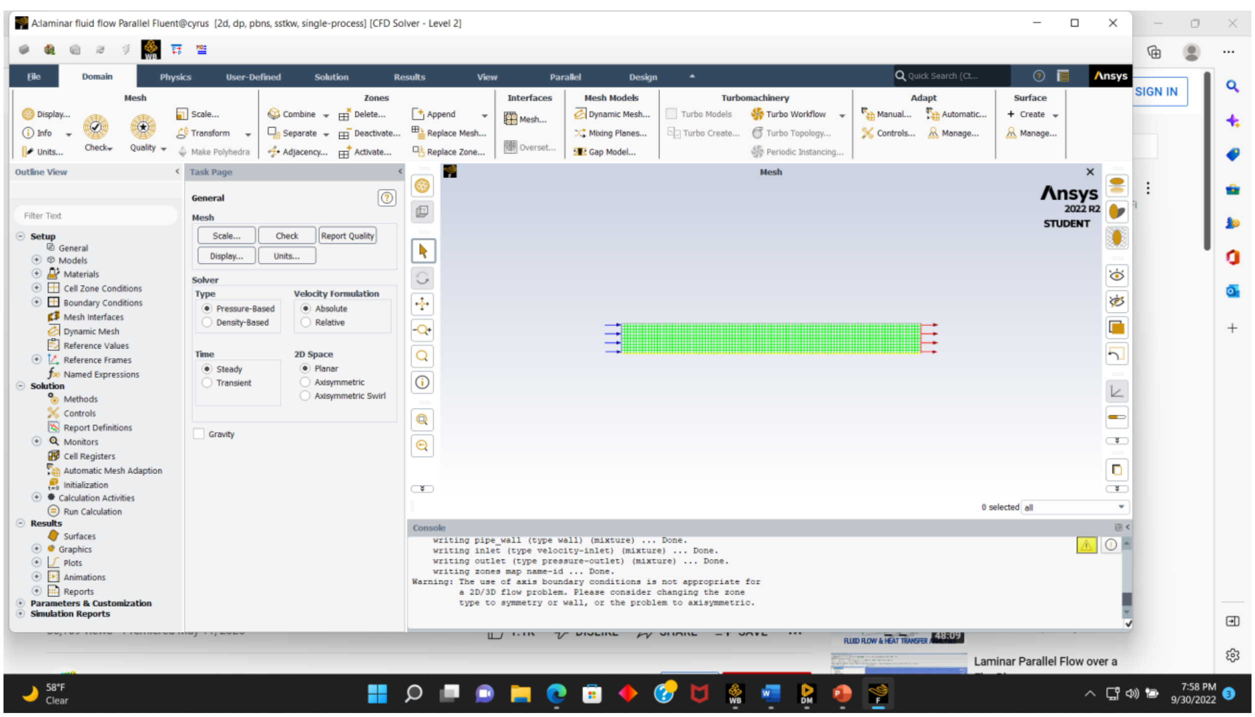




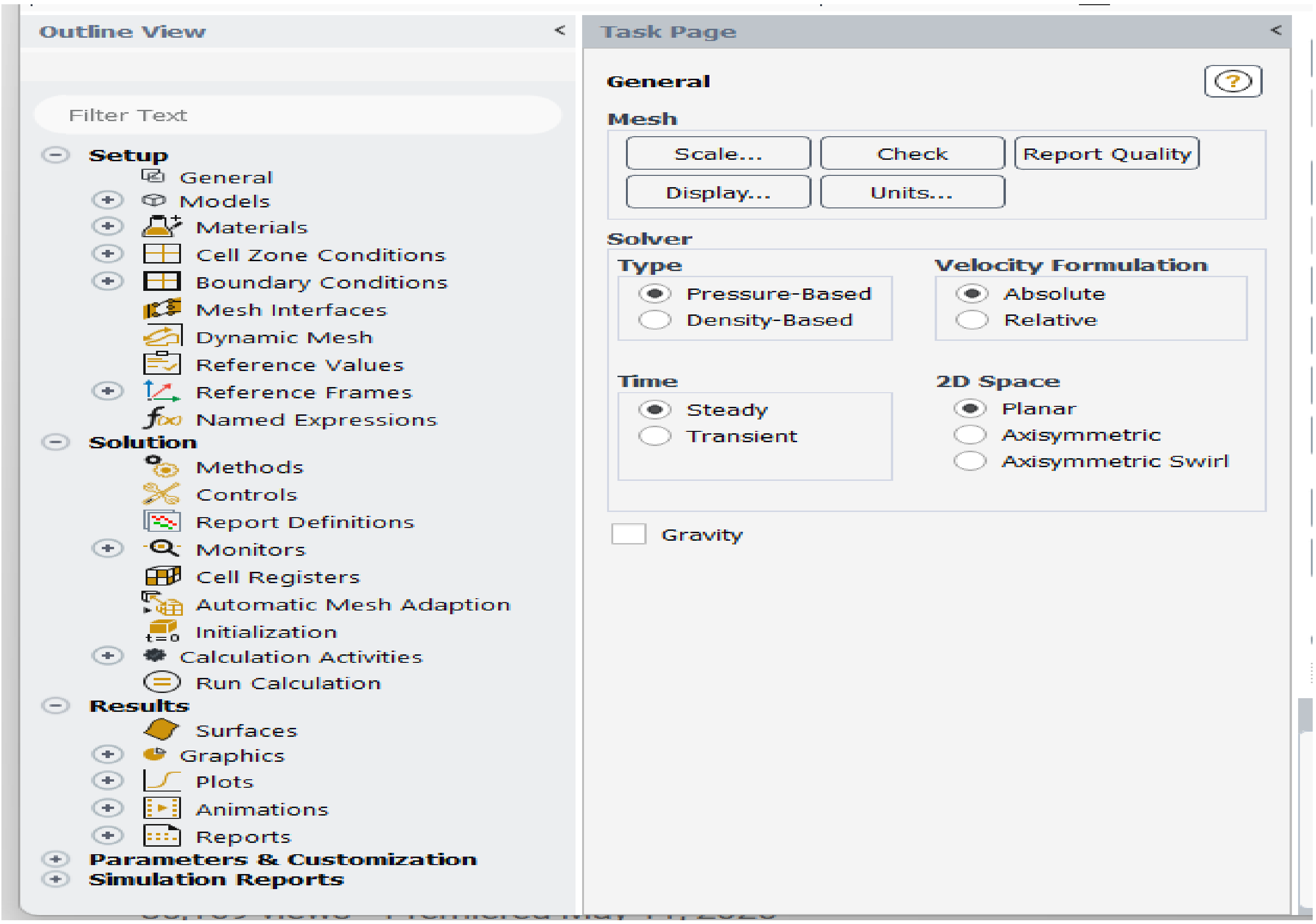
Click “Double Precision” and then click “Start”



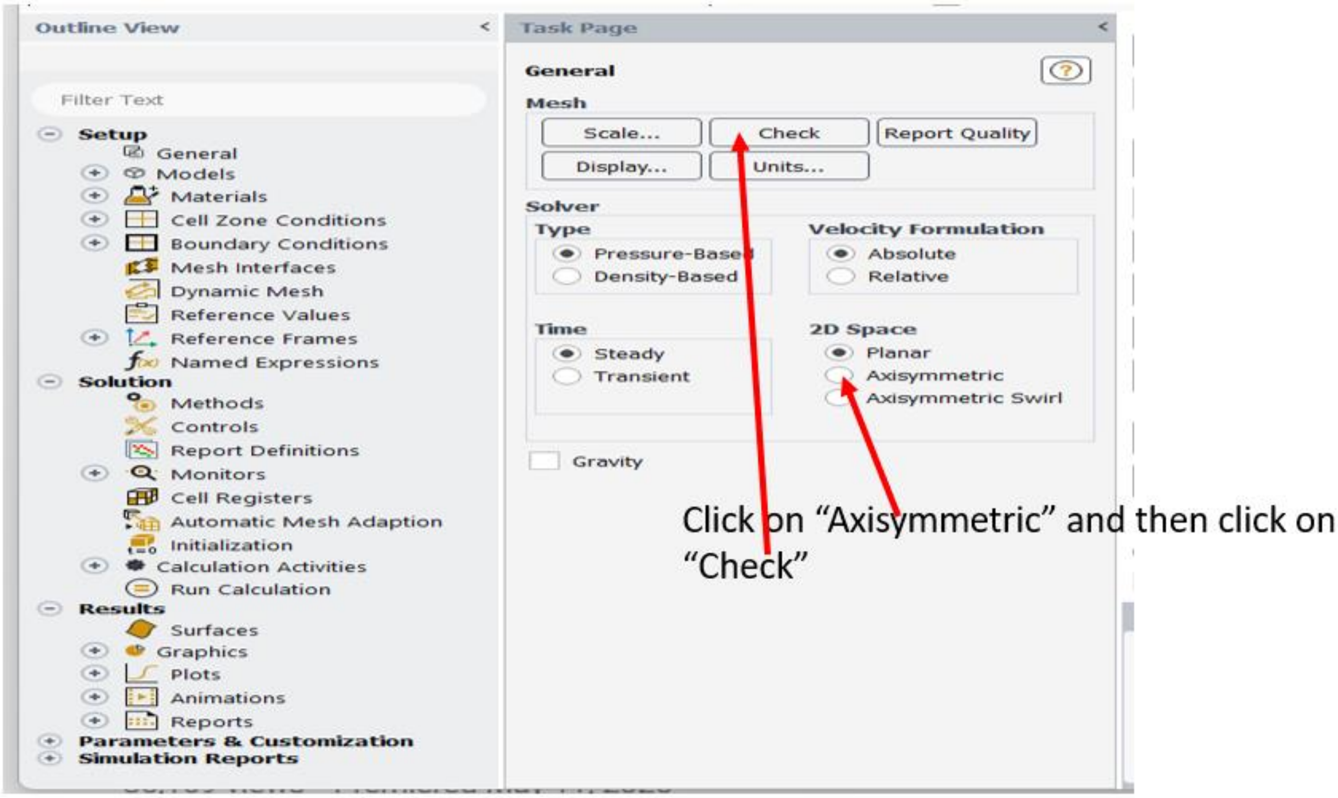
The following window appears. This is a new window and is defined as ”f” at the bottom of screen.



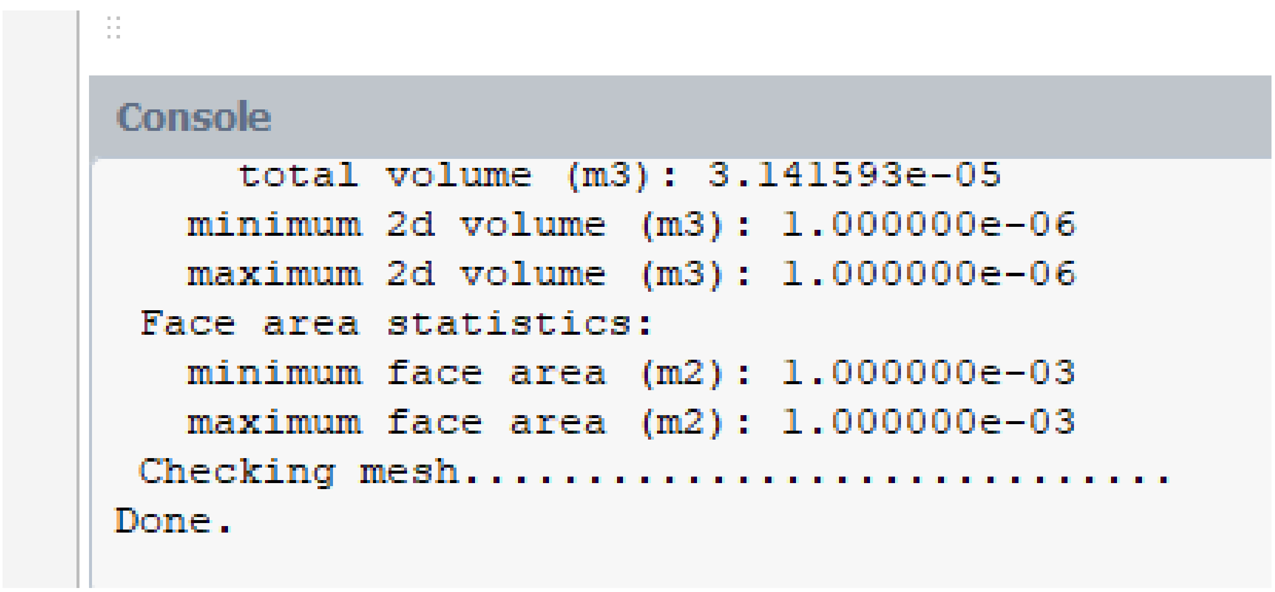
The following is the above window zoomed in on the left.



If check is clicked with above choices, there is a warning on the console. Change the 2D space to “Axisymmetric” above and then click on check as shown before. Always look for errors and warnings on the console window. Address all warnings and errors before proceeding.

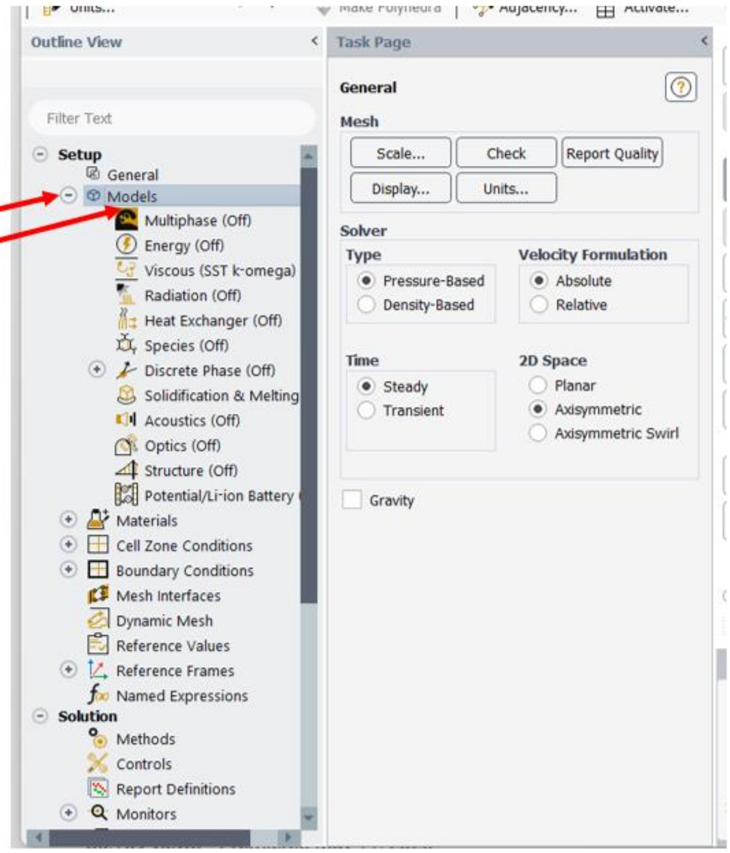


At the bottom of "f" screen, under console, the result of the checking is displayed as shown below.

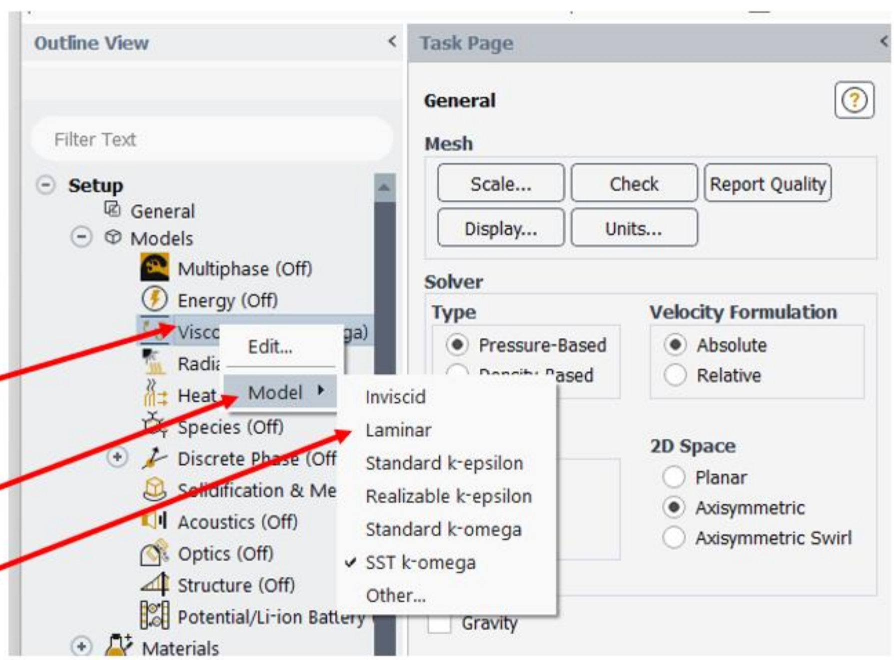


The above shown section is the bottom of "Console" section. The console message is long, and it can be read by scrolling up and down in the "Console" section.

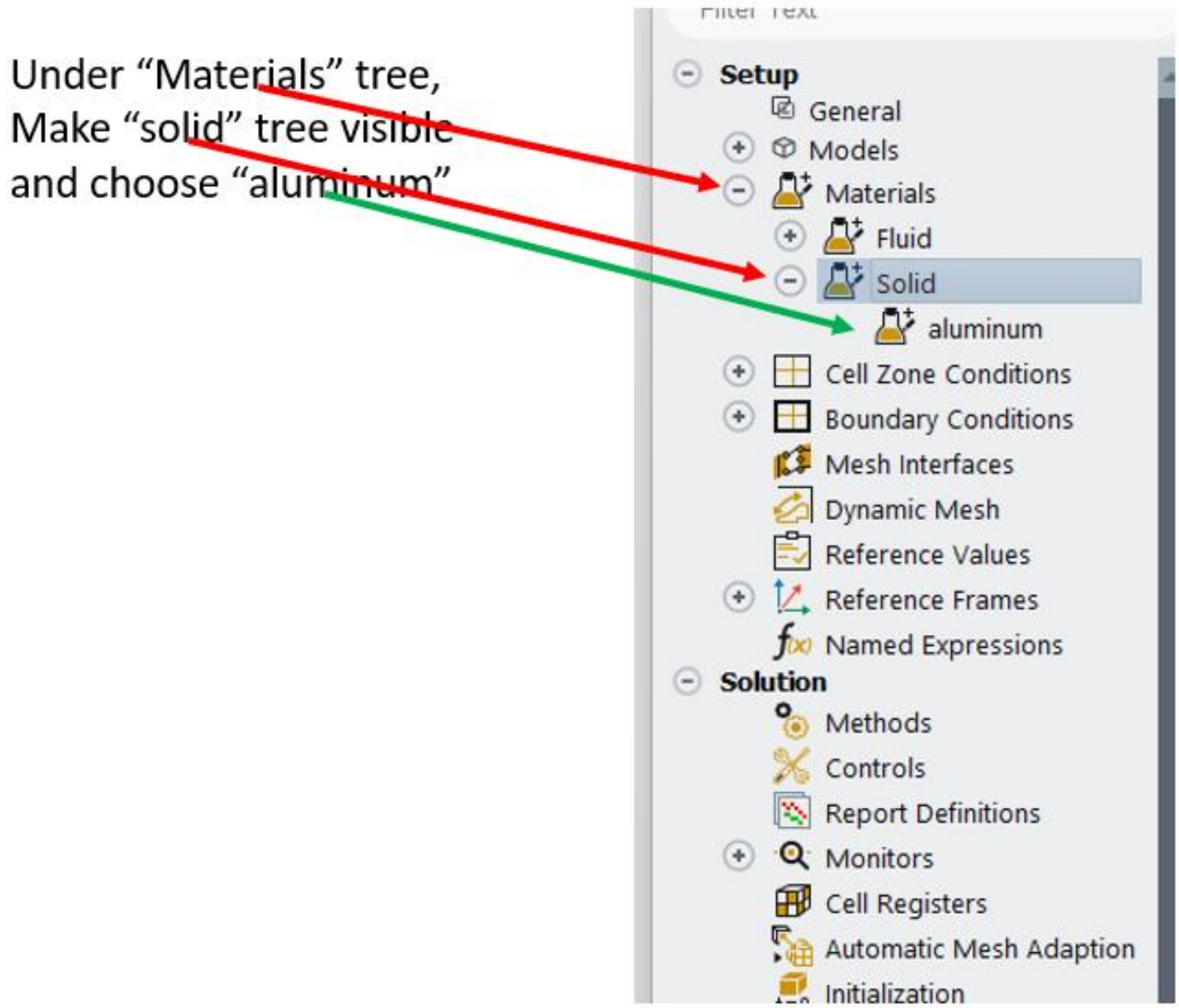
Click on the "+" next to "Models" to make the model choices visible



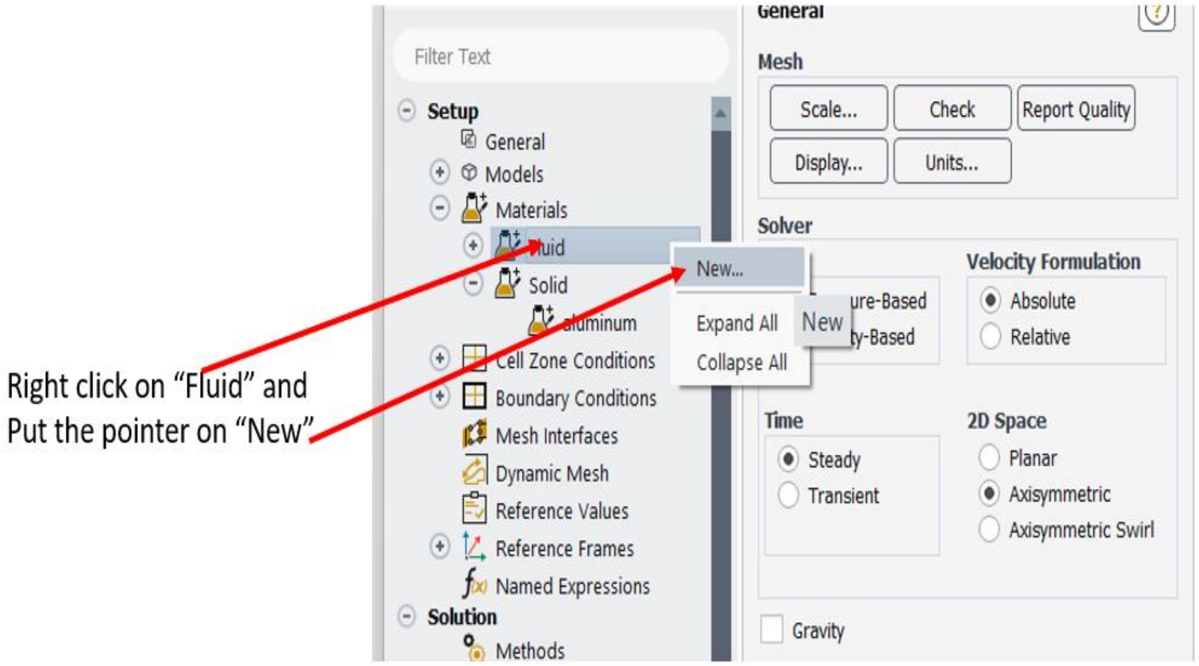
Right click on "Viscous" and then Put the mouse pointer on "Model" And then click on "Laminar"



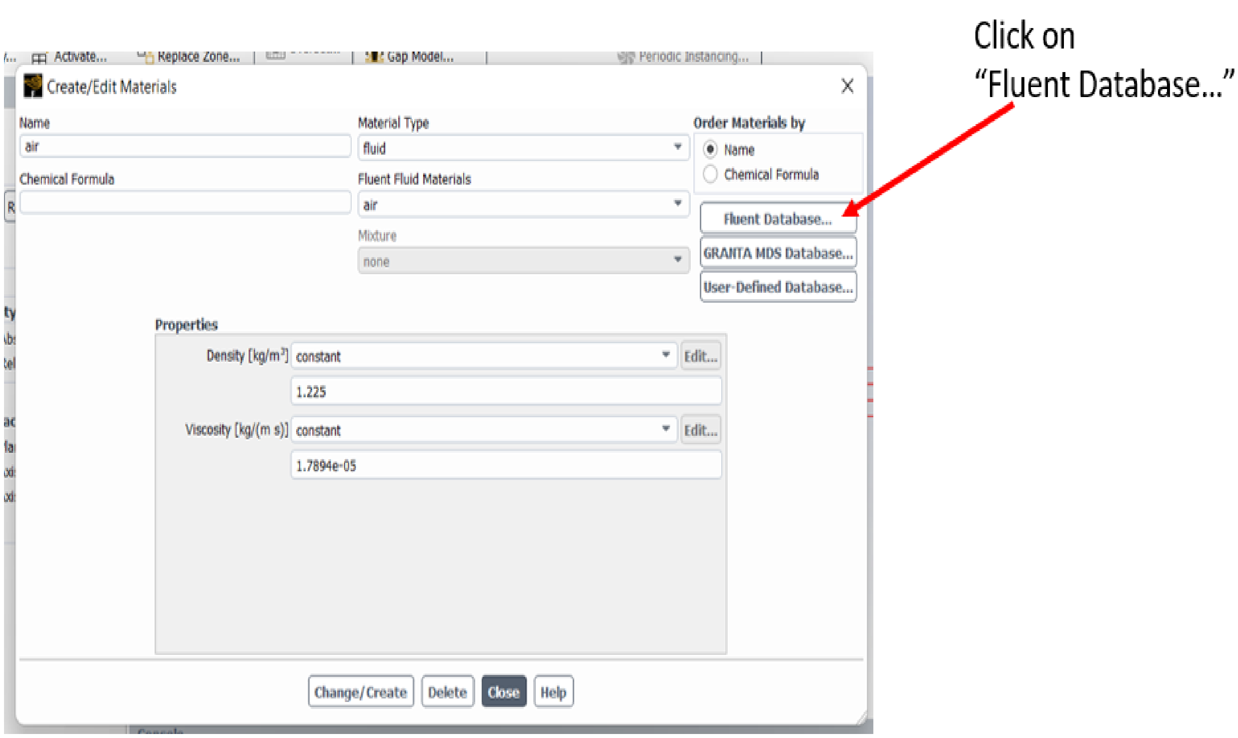
Click on the “-“ next to “Models” on the tree on the left of the “f” window in order to make the choices associated with “Model” become invisible in order to open up room on the screen. After closing the “Model” tree, click on the “+” next to “Materials” in order to make the “Materials” tree visible.



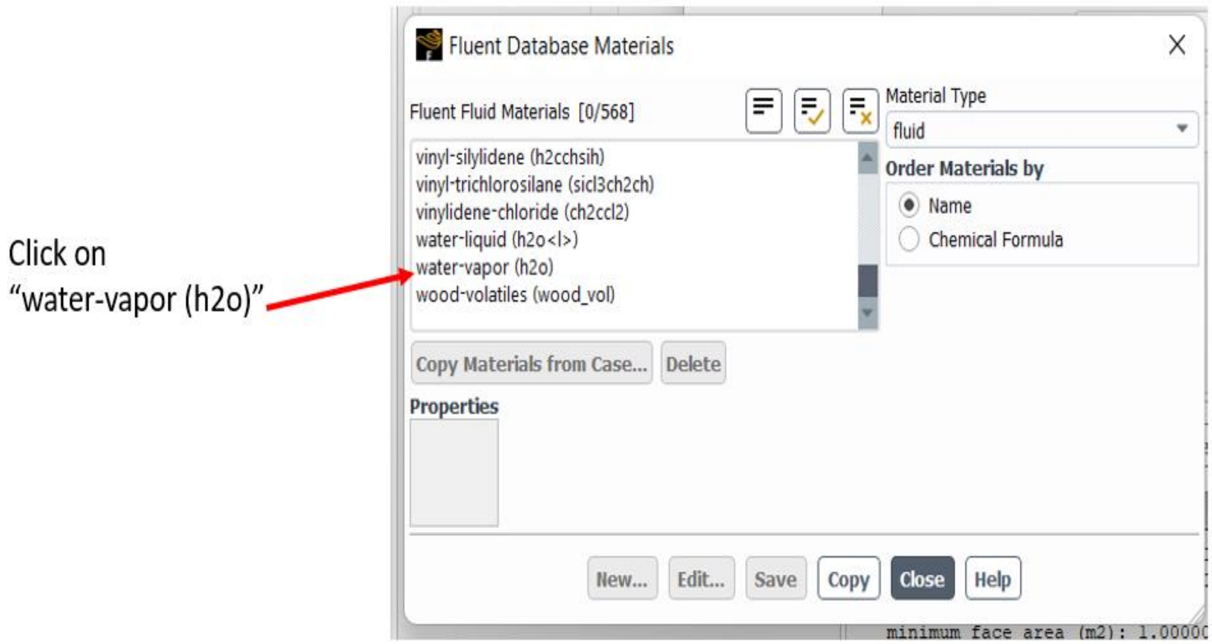
Make the “Fluid” tree visible.



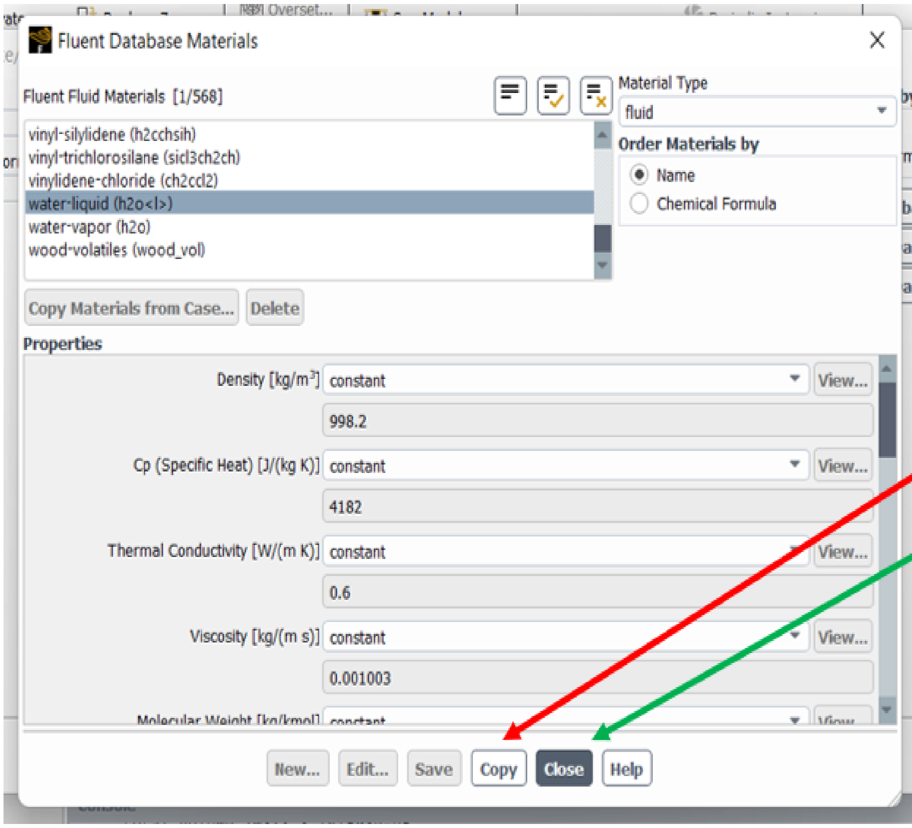
Click once the pointer is on “New” and the following table appears.



“Fluent Database Materials” dialog box as shown blow appears. Scroll down until “water-liquid (h20<l>)” can be selected.

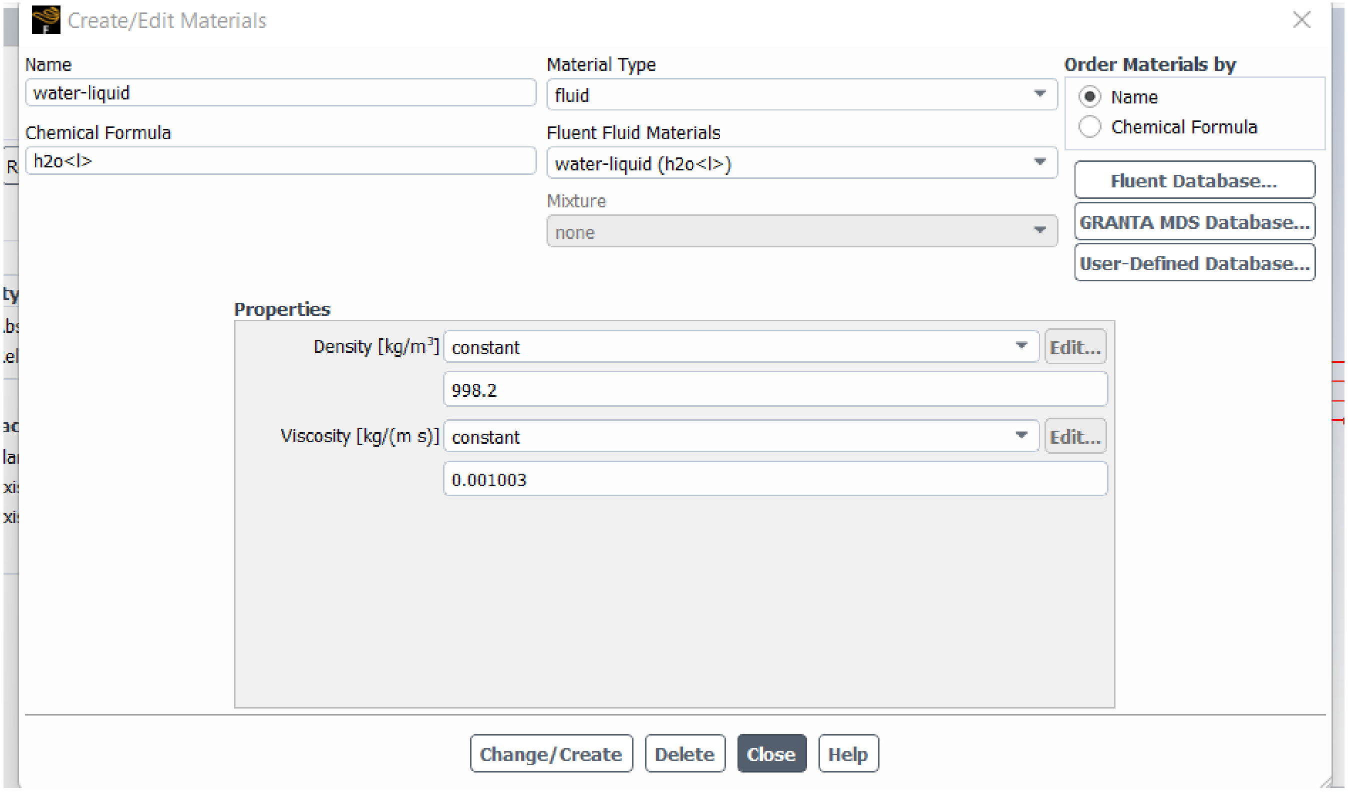


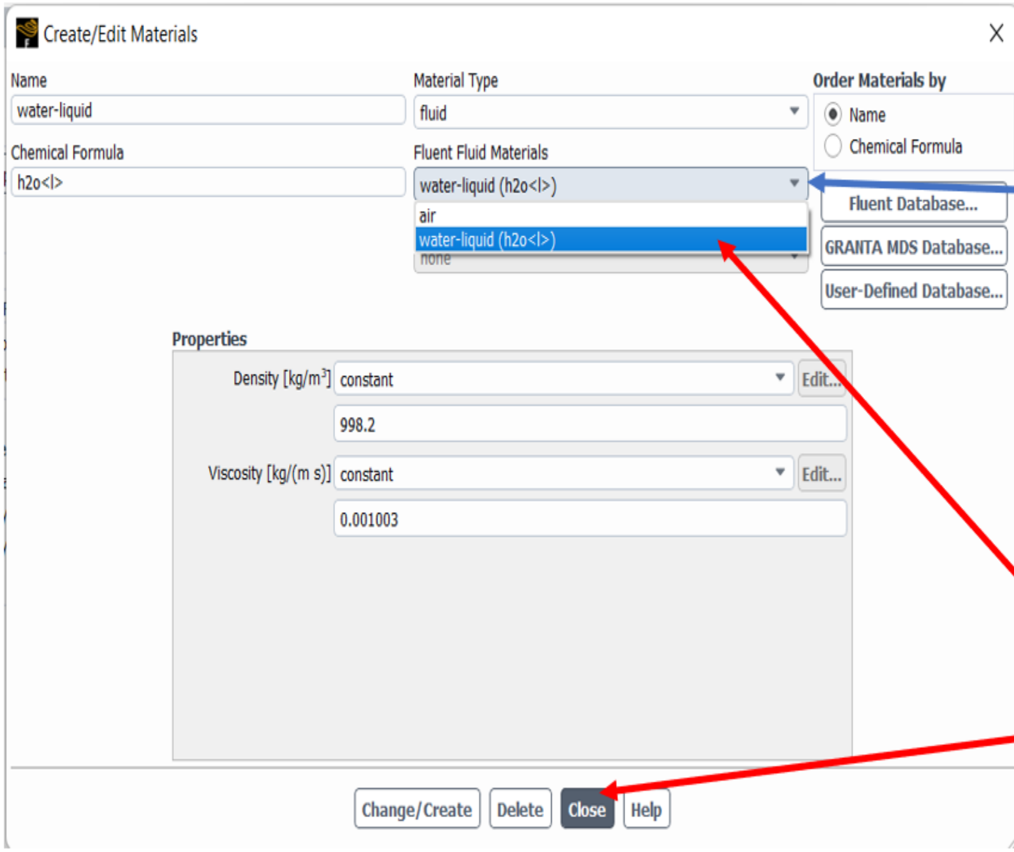
Once the selection is made, the “water-vapor (h2o)” properties appear as shown below.



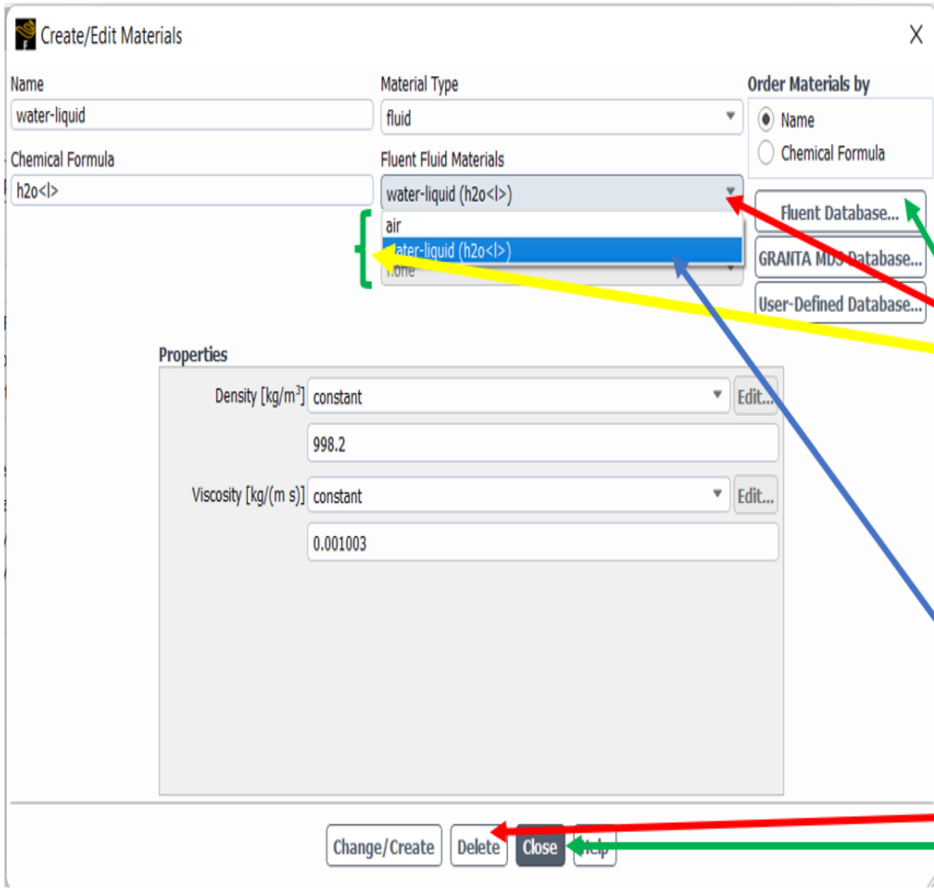
Click on “Copy” and then “Close”

The following window appears where it is shown that the water properties are selected for fluid.





Clicking on arrow Makes all fluid material types available to the model visible. The material that is to be included in the model must be highlighted and then "Close" clicked.



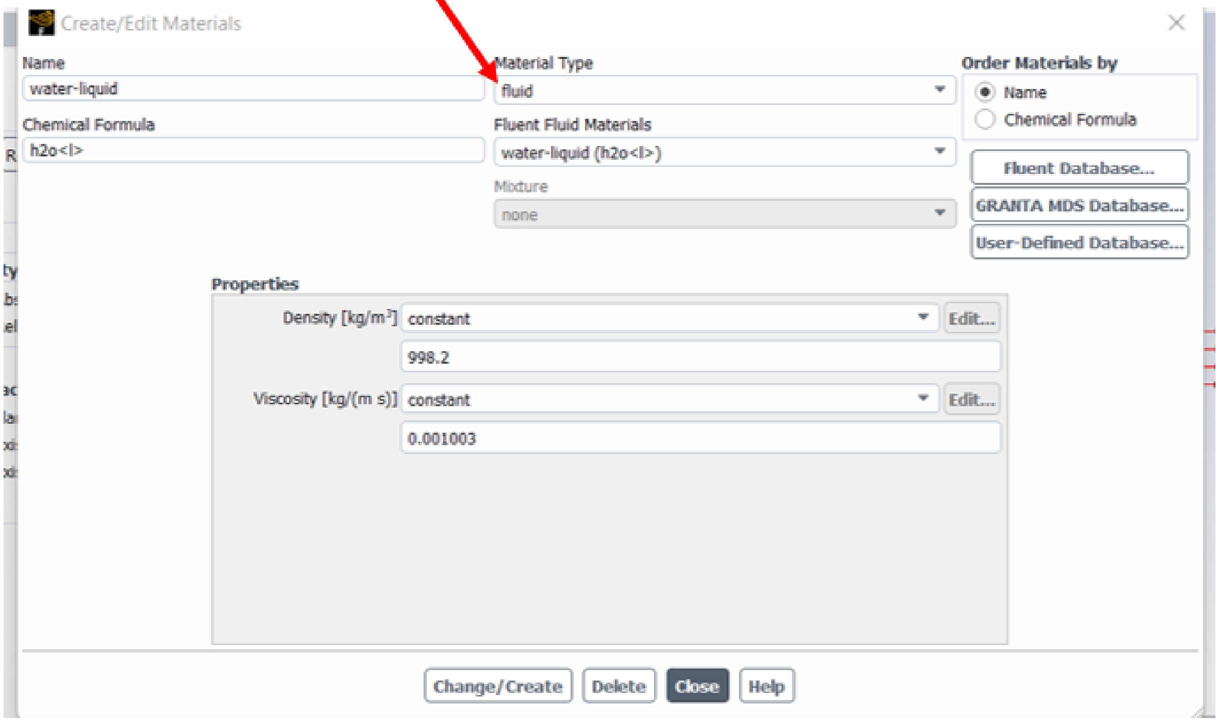
Clicking on arrow.

All the available materials that have been copied from "Fluent Database" become visible.

In order to delete a material,

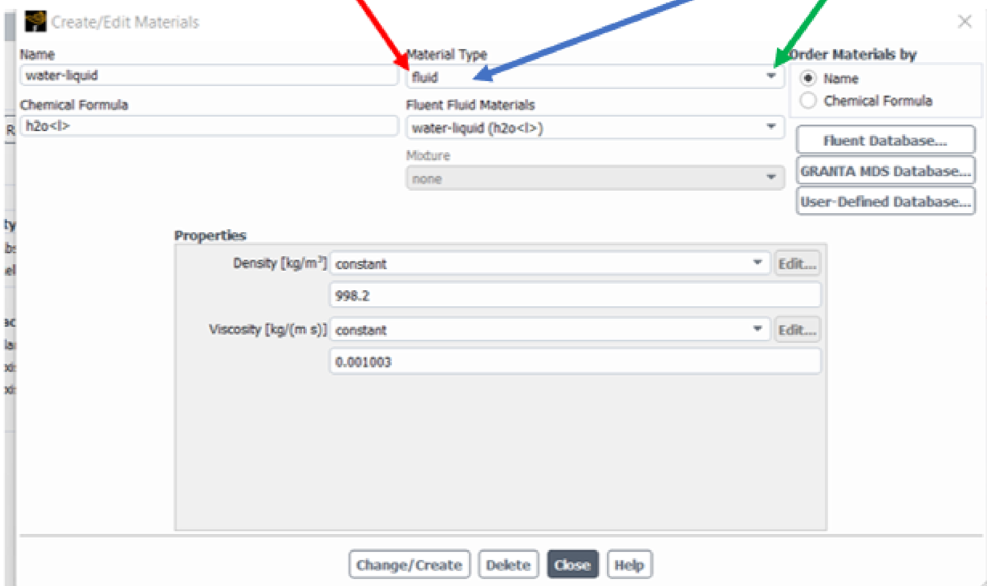
Highlight it, and then click in "Delete" and then "Close"

Note that the properties shown are for the fluid.

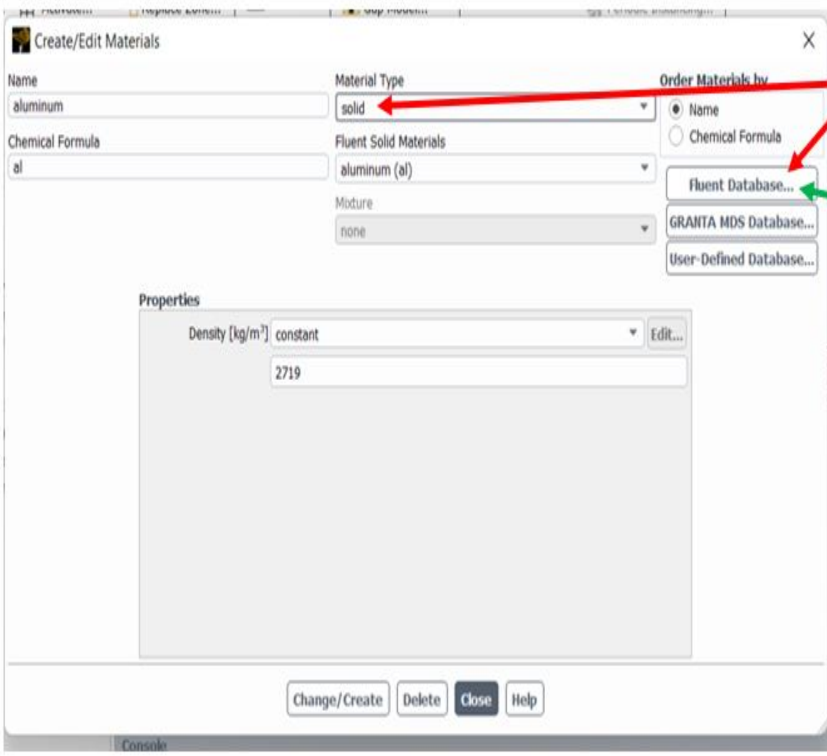


Click on this arrow, and change the selection from fluid to "solid"

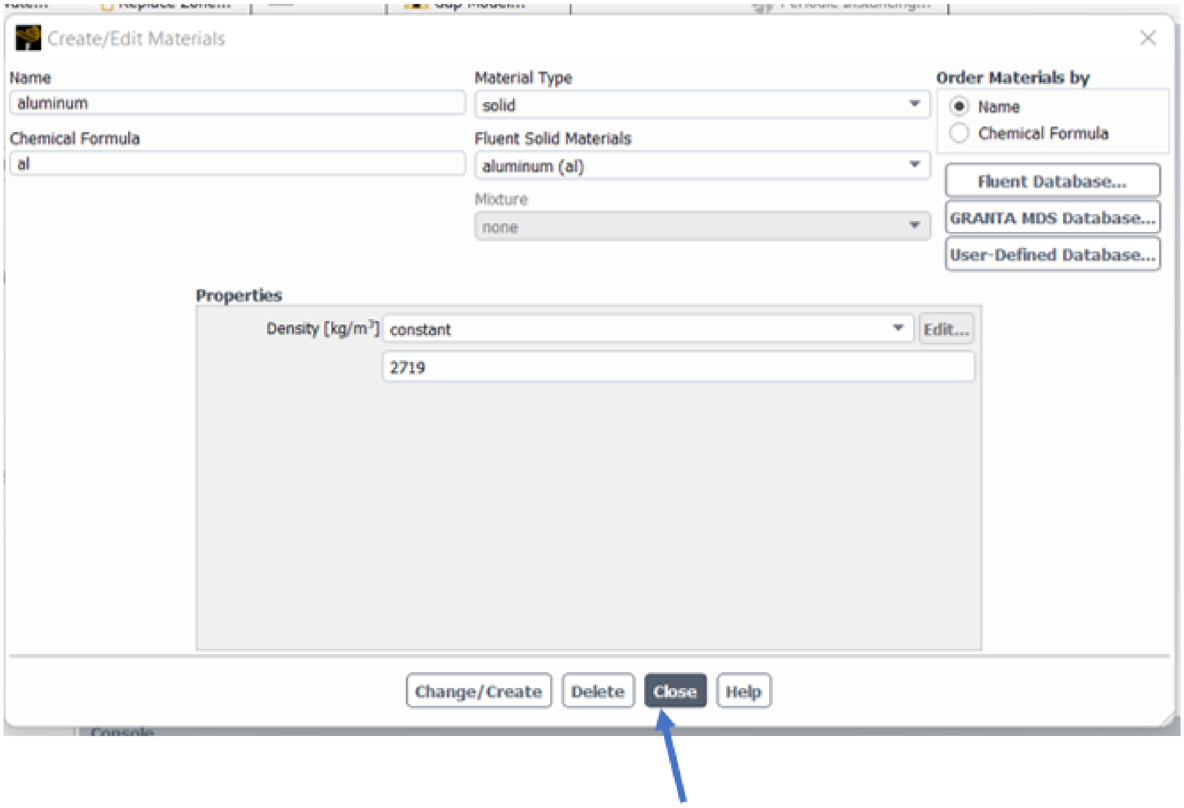
Note that the properties shown are for the fluid.



The solid material properties and the fluid material properties are all on the same list in the fluent data base. Choose aluminum for solid. By the selection of aluminum for solid and water for fluid, the model is going to simulate water flow in an aluminum pipe.

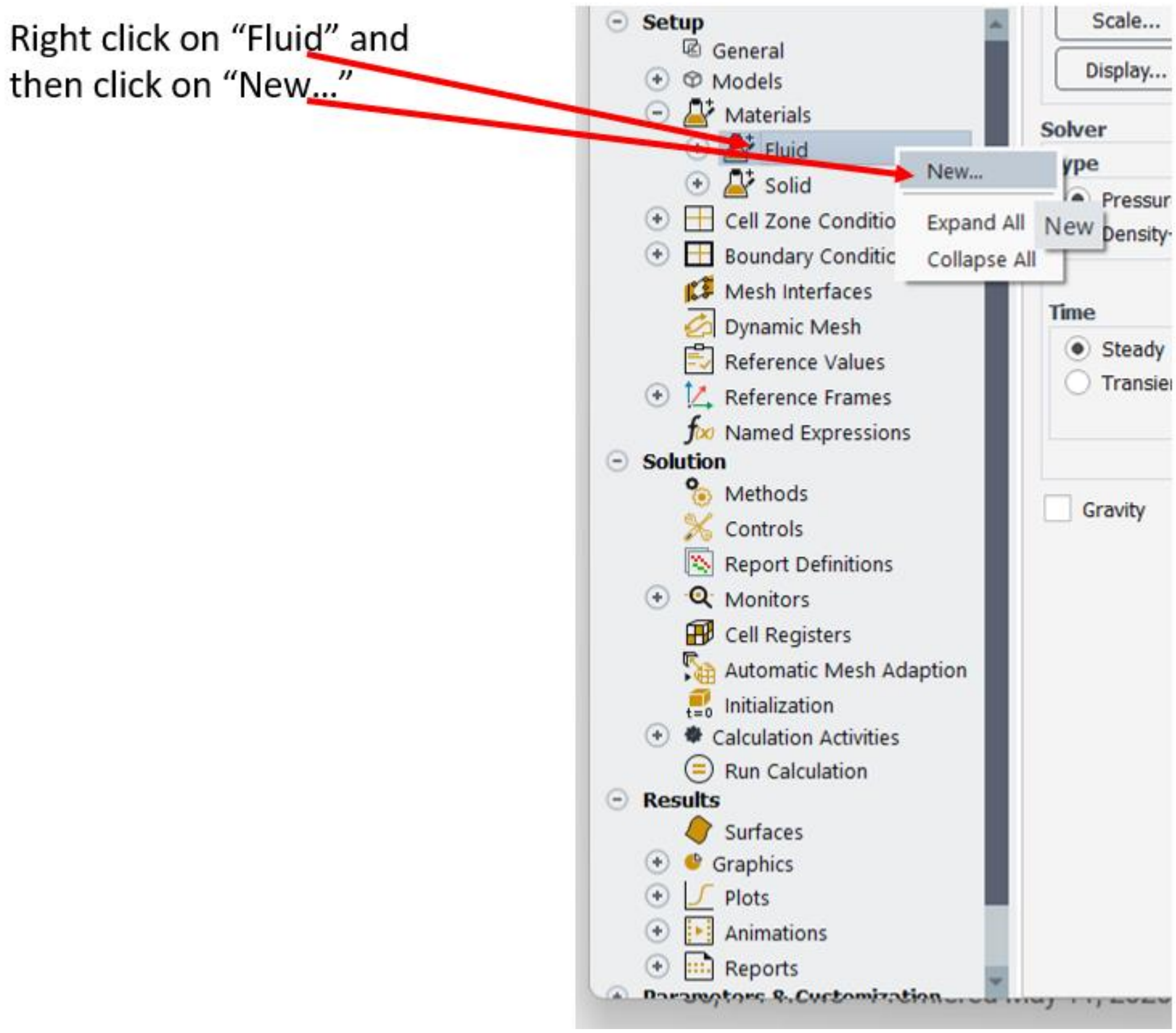


Similar to the technique used for selecting the fluid properties from the "Fluent Database...", the solid material properties can be defined from the "Fluent Database..."



Click on "Close"

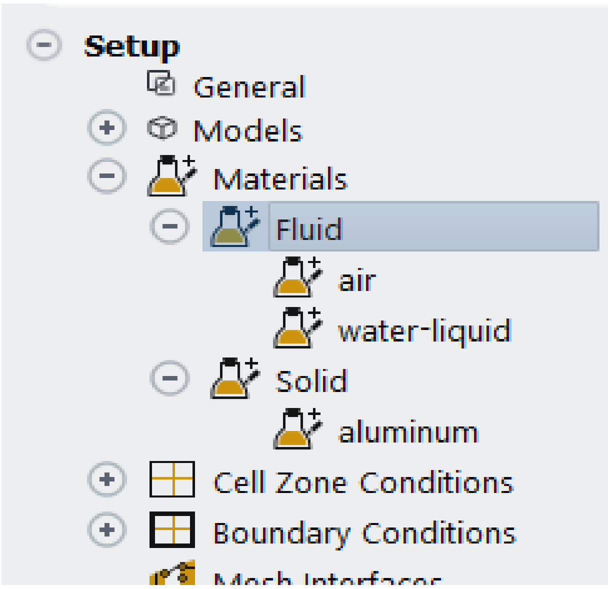
At this point, the pipe material and the fluid material are available. The fluid properties will be assigned to the mesh. The procedure for doing so is described later in this document.



As shown below, the only available solid is aluminum.

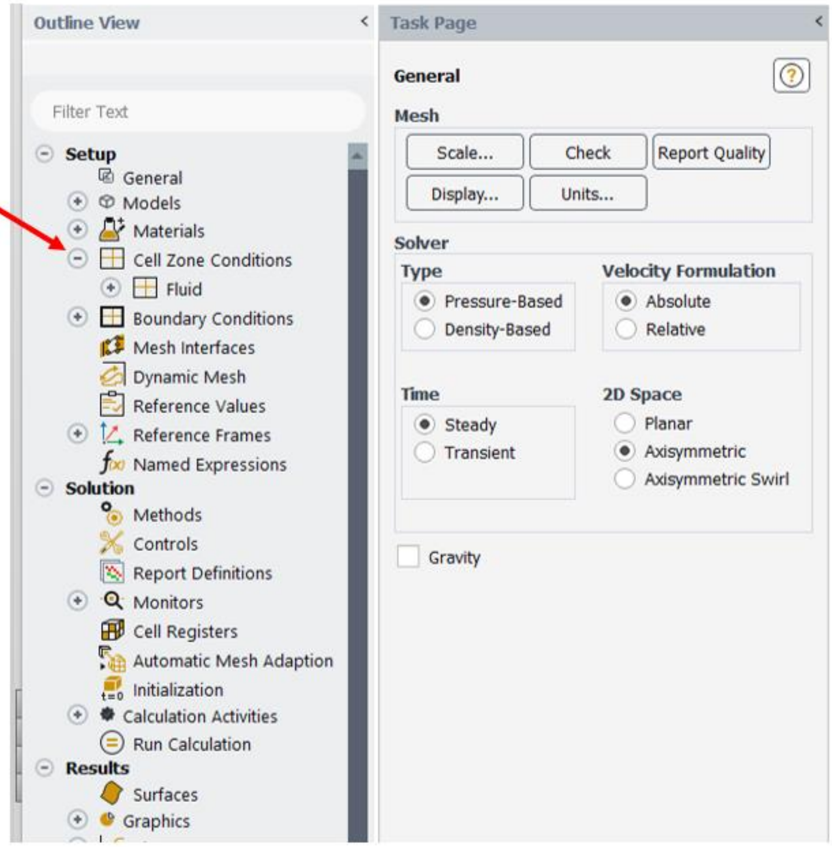
Click on “Close” above, to make the “Create/Edit Materials” window to disappear.

The following shows the material tree on the left of the “f” screen. It can be seen that the fluids are “air” and “water” and the solid is “aluminum”. I don’t know why “air” is there. I have not put it there.

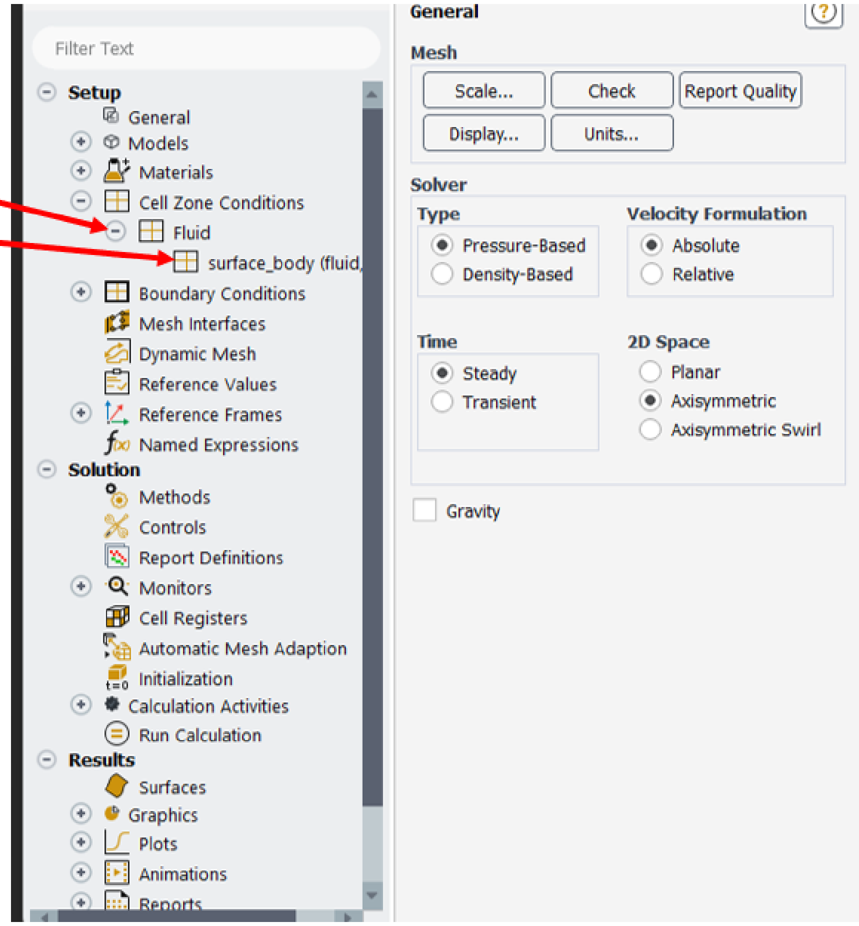


Close the “Material” tree by making the “-“ into a “+”. The screen appears as shown below. The next task is to assign the fluid to the meshed body.

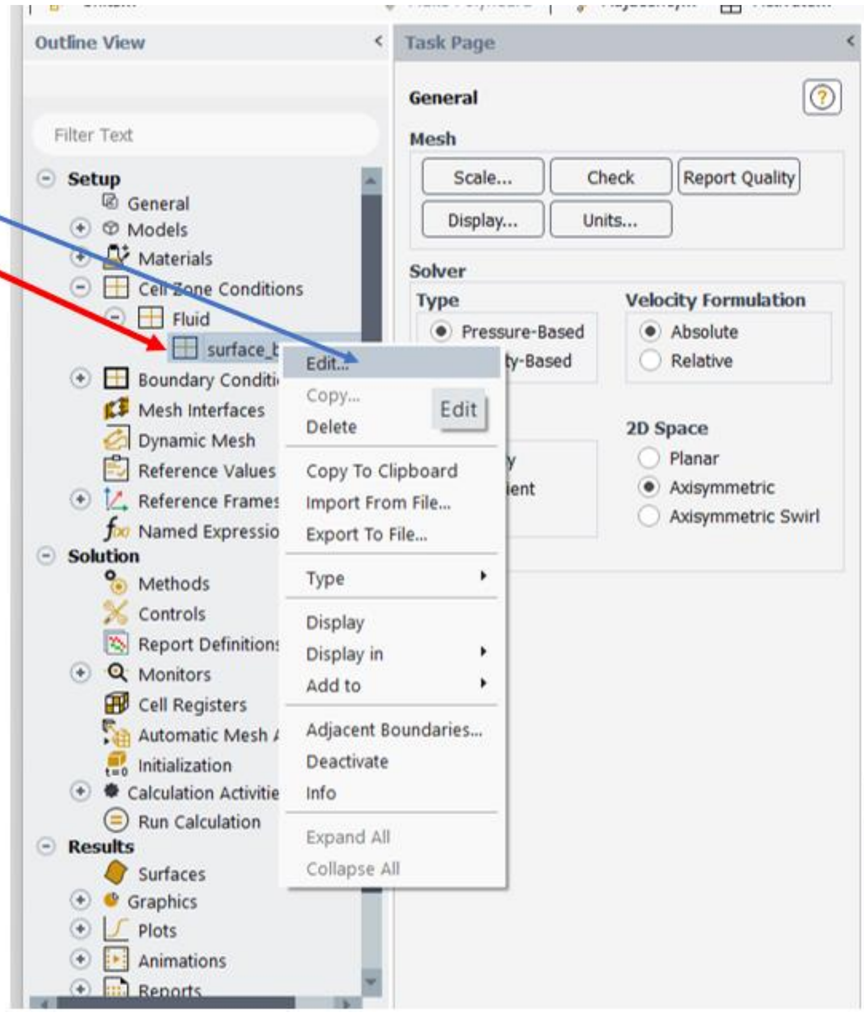
Click on this “+” to turn it into a “-” in order to make the tree visible.



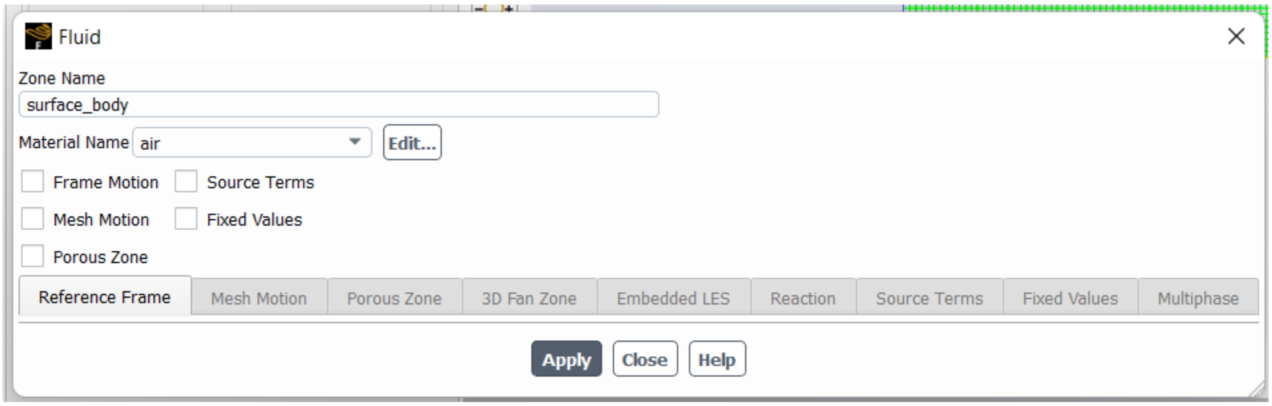
Click on "+" to turn it into "-"
In order to make "Surface_body (fluid)" visible

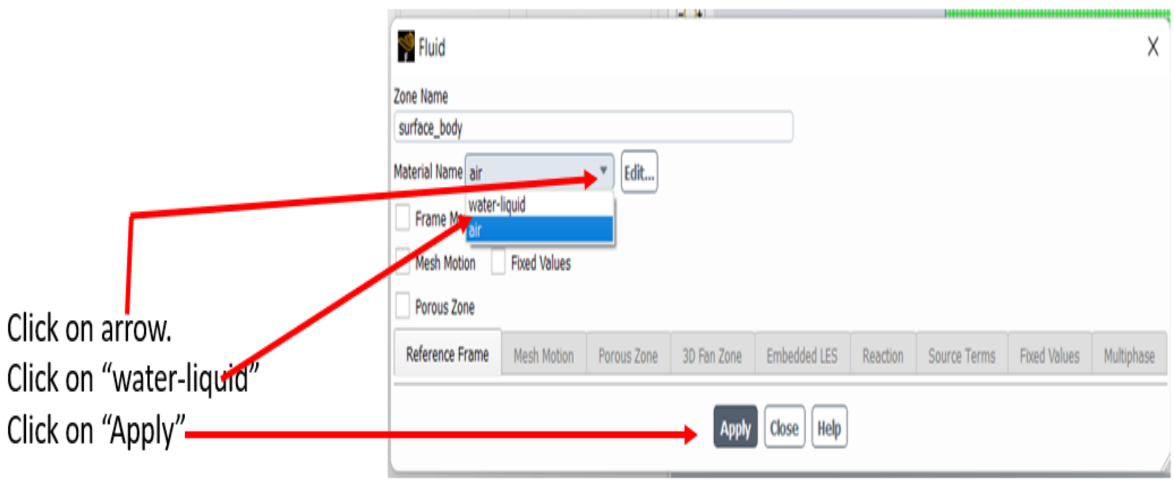


Right click on
"Surface_body (fluid)"
And then click on "Edit..."



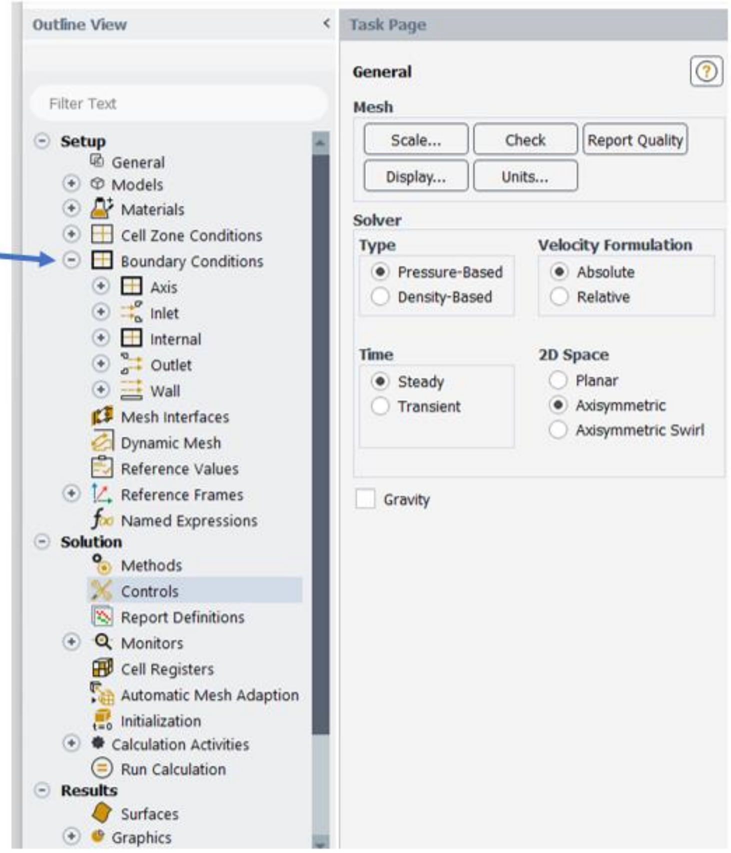
The following dialog box appears.



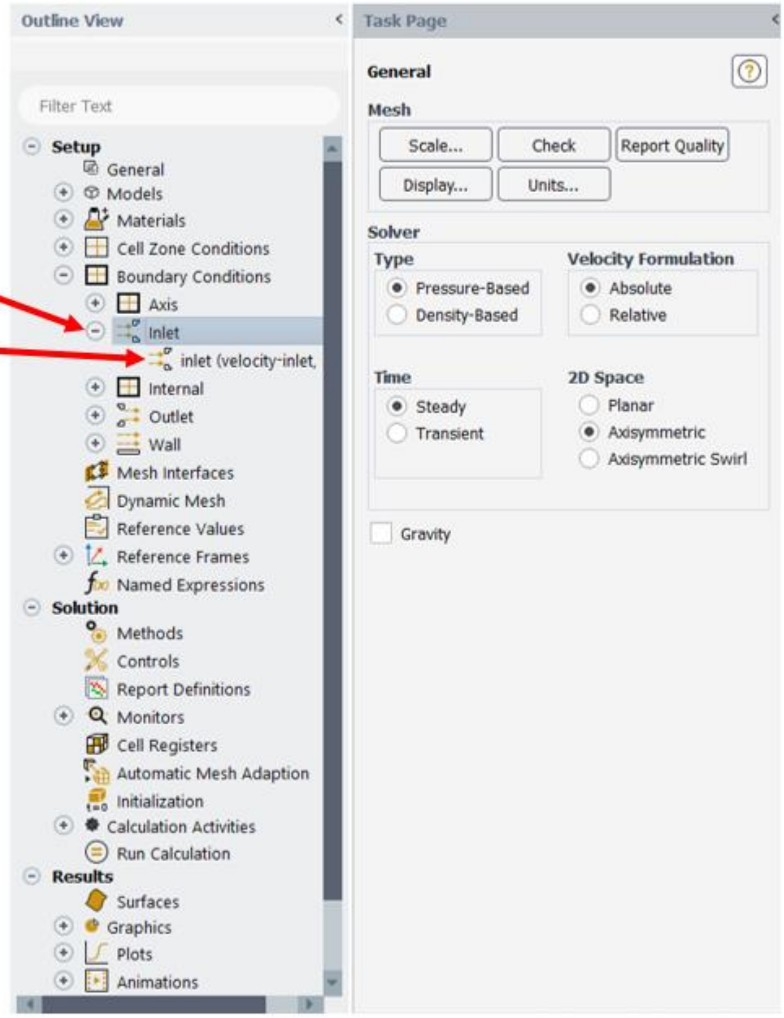


Click on "Close" after selecting "water-liquid".

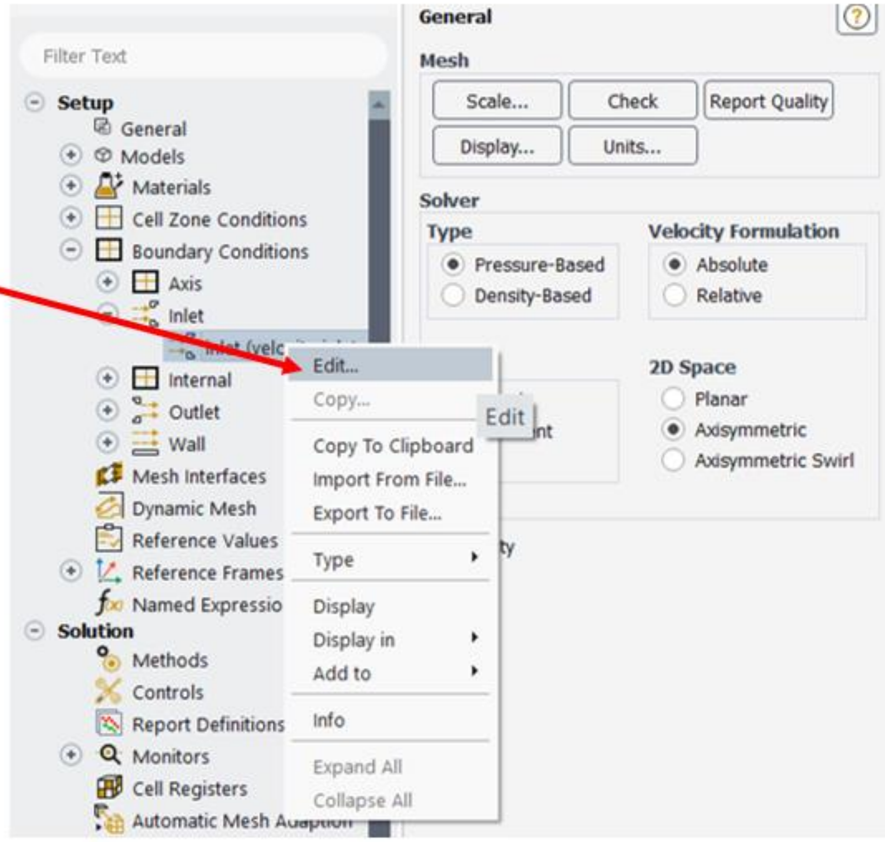
Make the Boundary Conditions" tree visible.



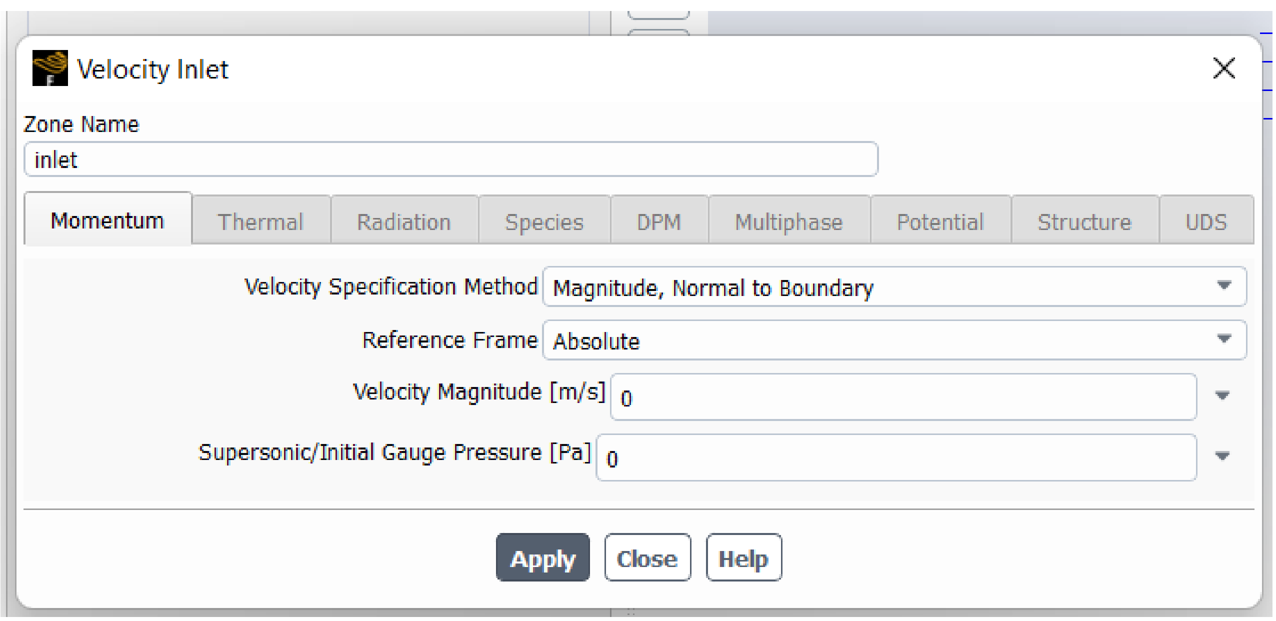
Make the "Inlet" tree Visible.
right Click on "inlet (Velocity-inlet,

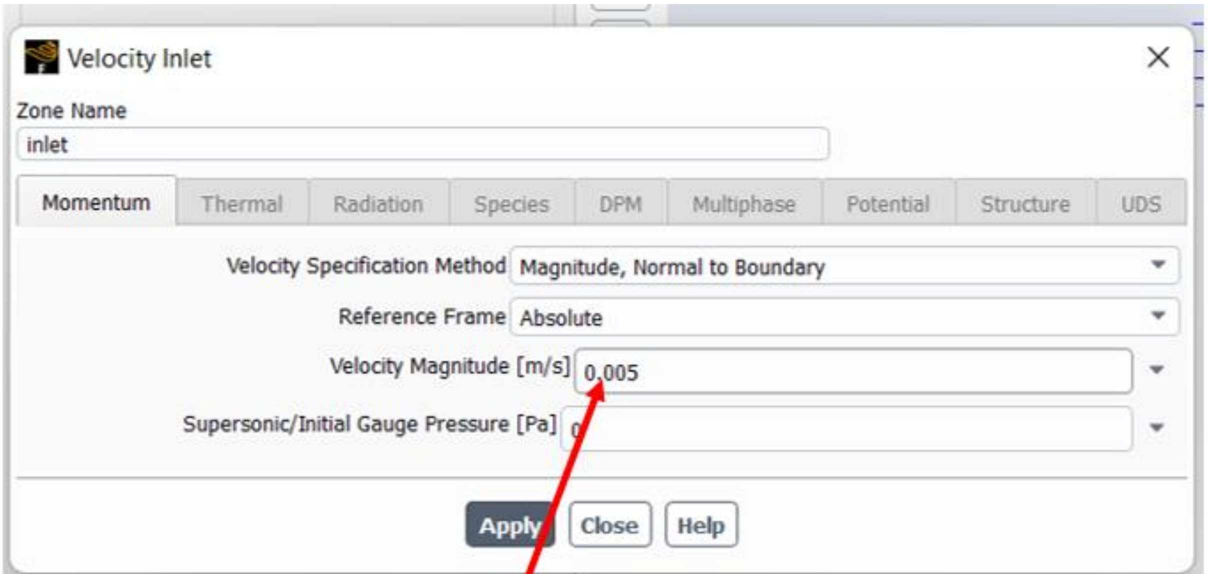


Click on "Edit"

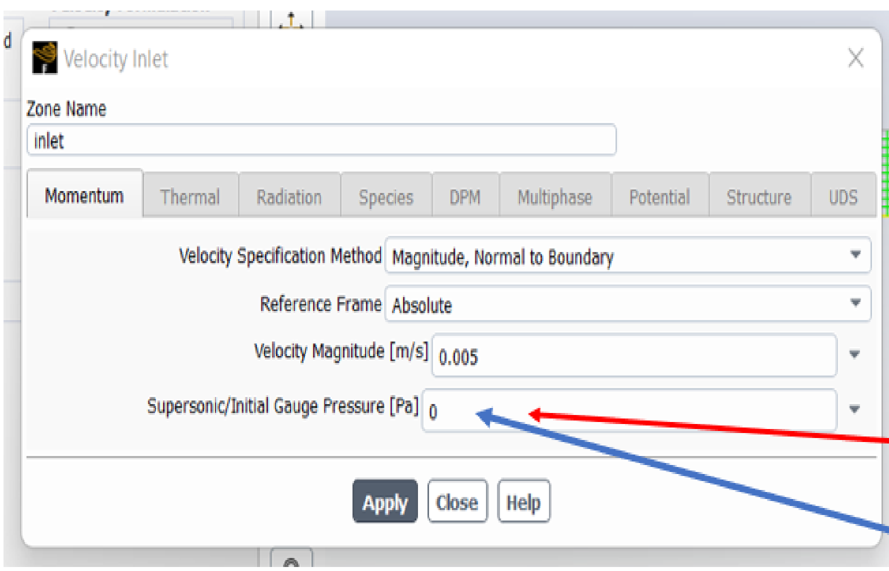


The following dialog box appears.





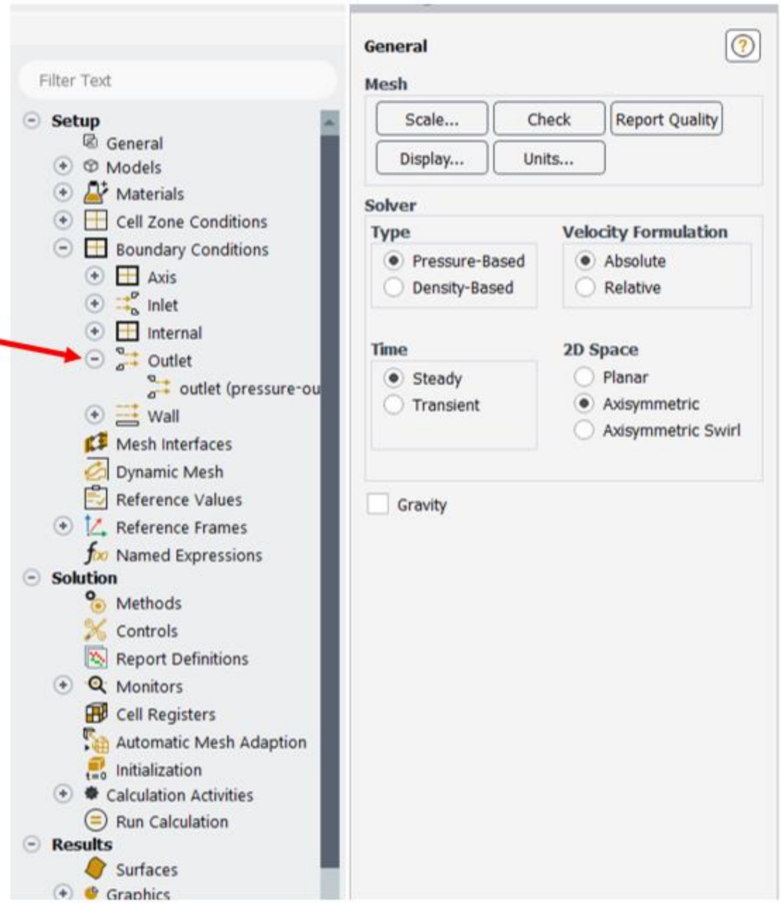
Change the velocity
to 0.005



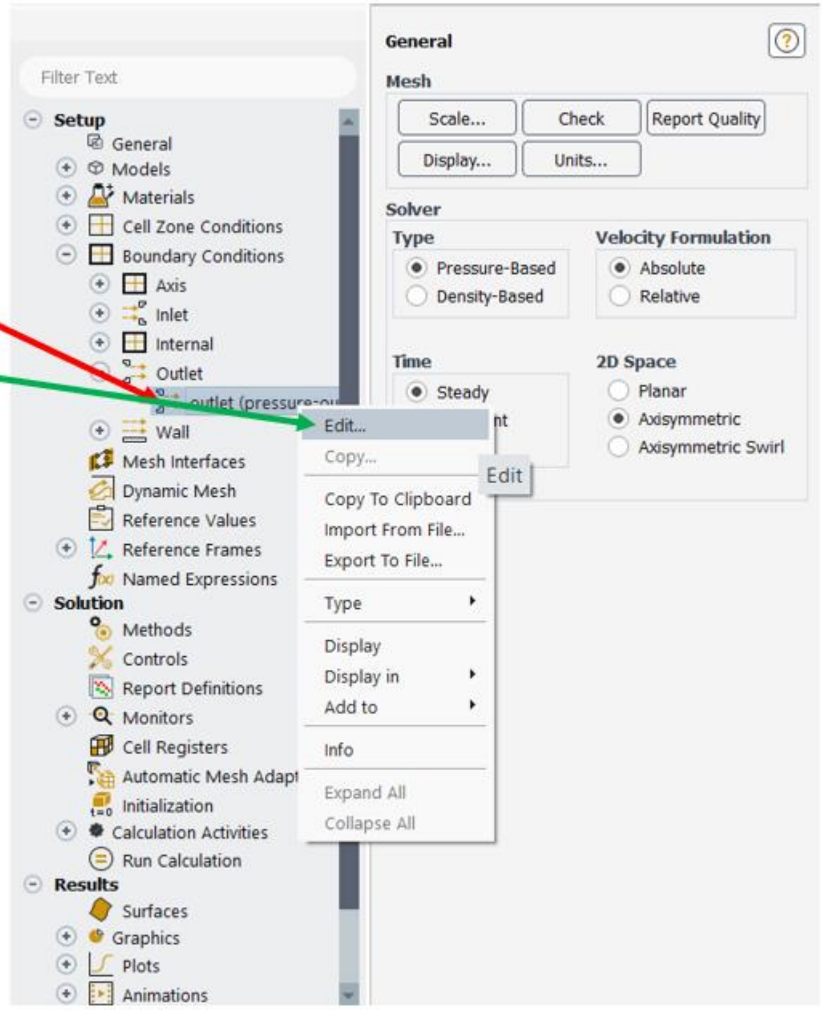
By having the inlet gage pressure as "0", the inlet pressure is atmospheric. This pressure can be changed here

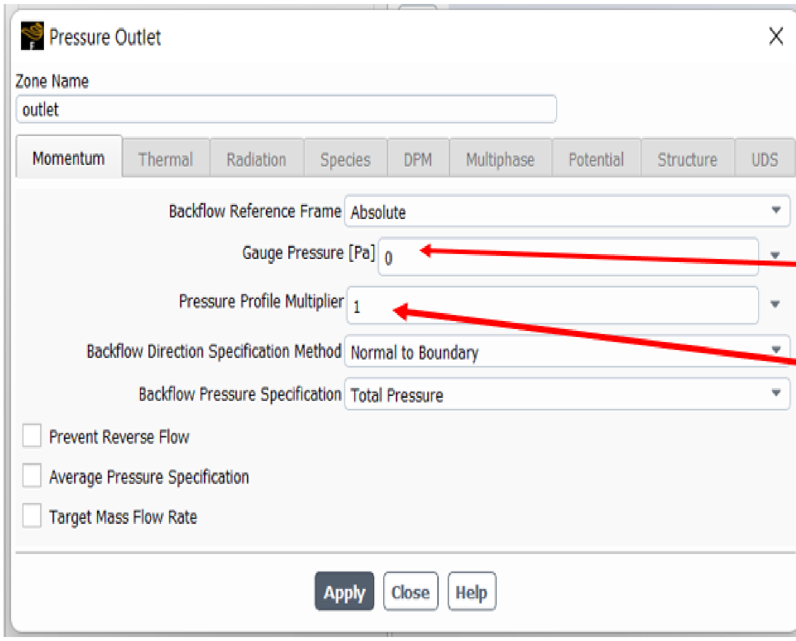
Click on “Apply” and then “Close” after the velocity and pressure selections are completed.

Make the outlet tree visible by changing the “+” to “-”



Right click on
“outlet (pressure-out)”
And then click on
“Edit”

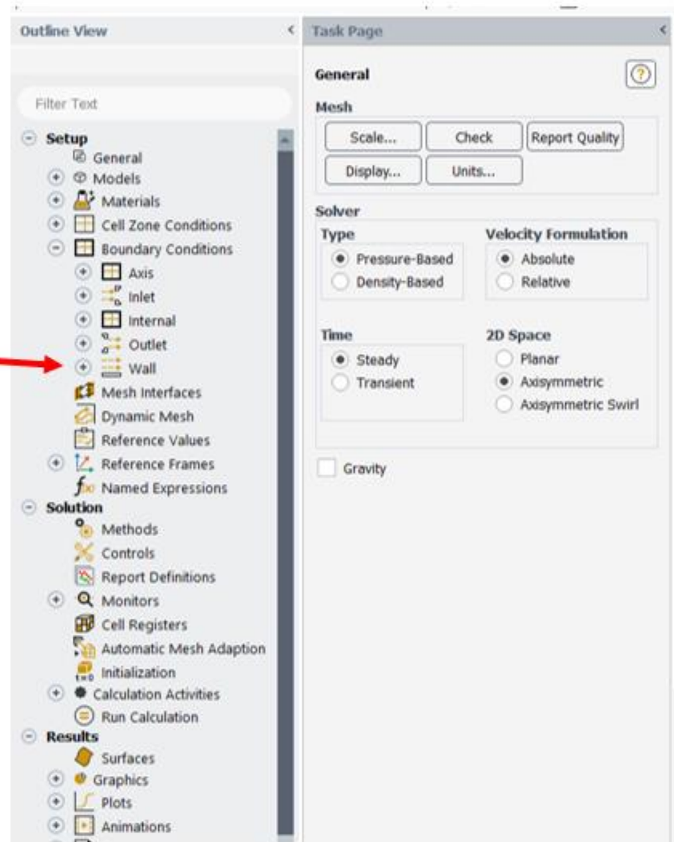




By keeping the gage pressure at 0
And the multiplier at 1
The outlet pressure is atmospheric

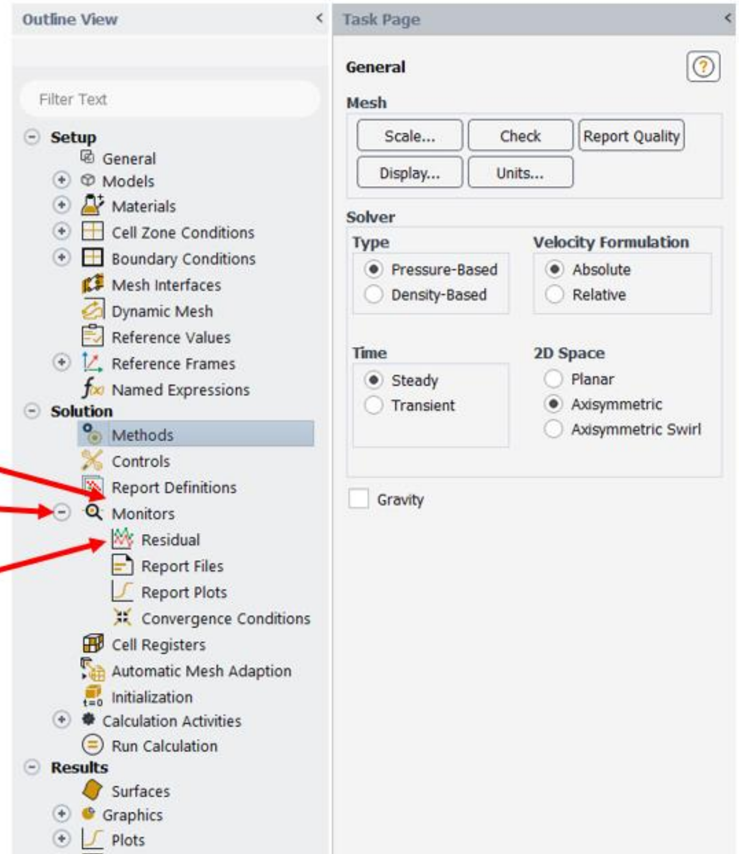
After making the above selections, click on “Apply” and then “Close”.

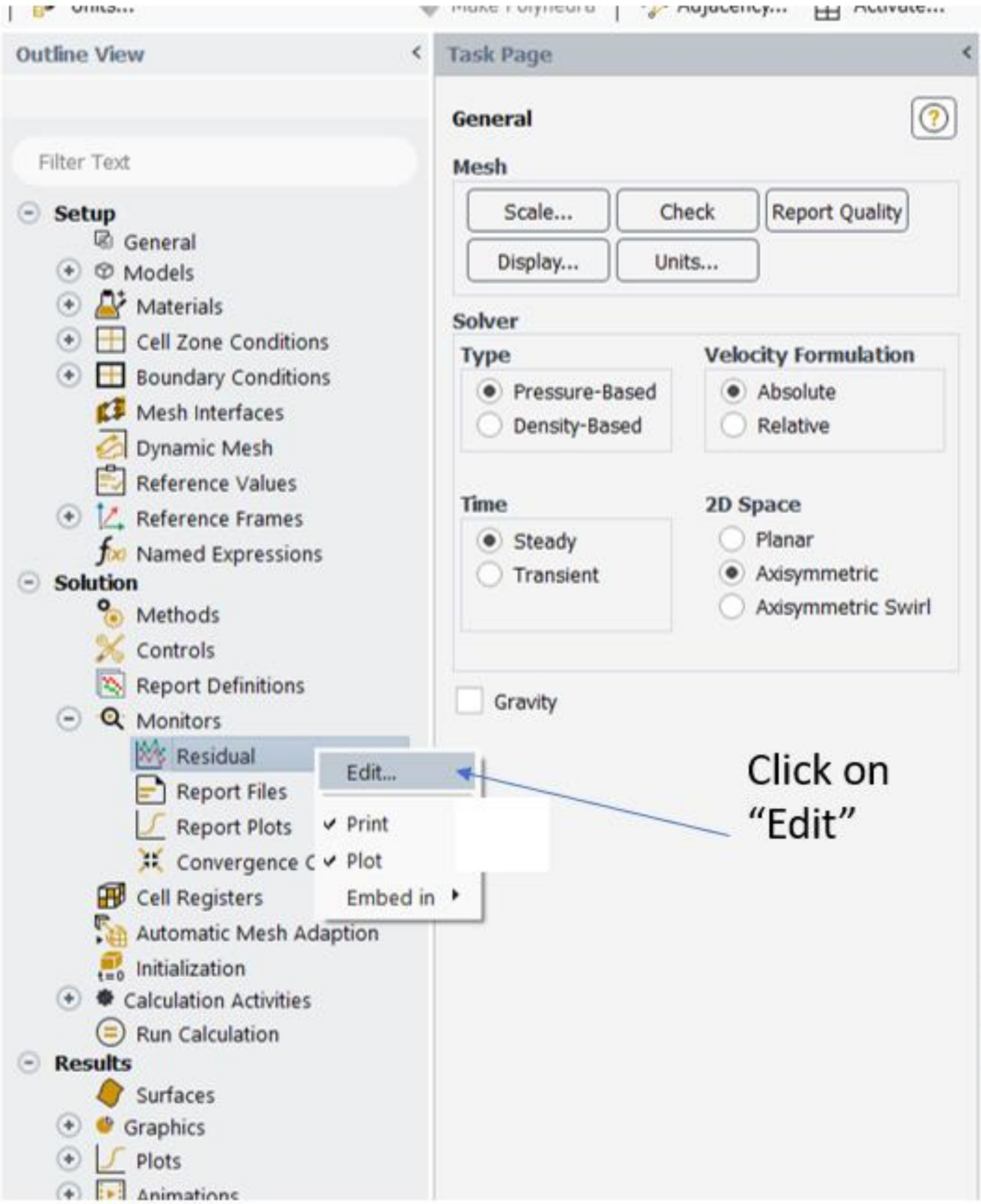
Because the pipe is stationary, the default "Wall" Conditions should be as they are

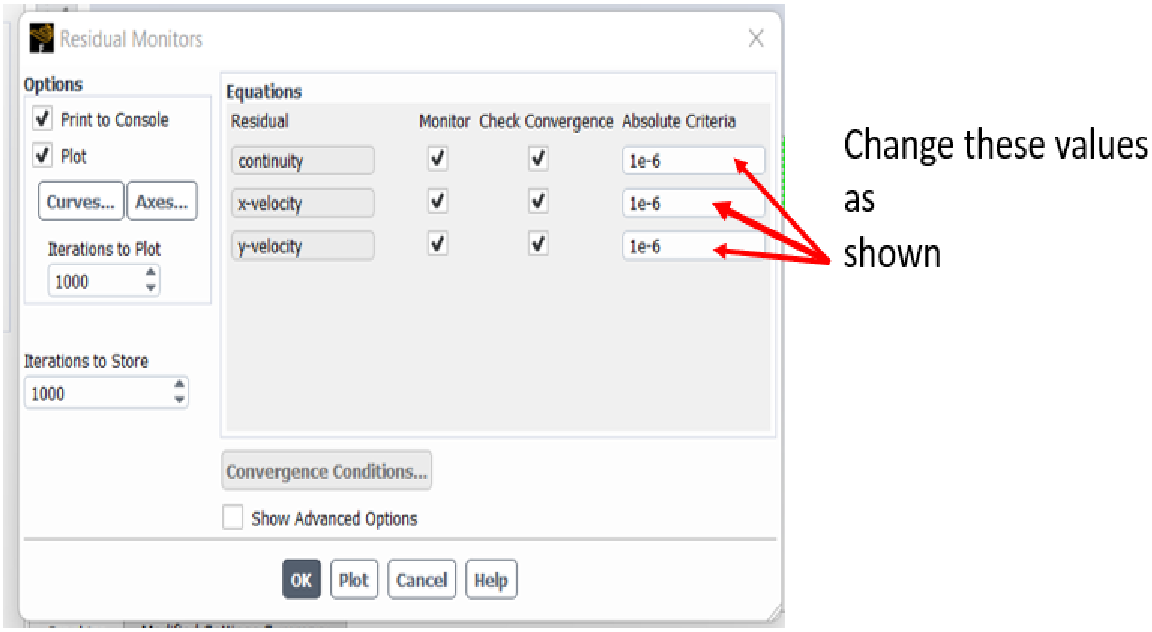


Make the "Monitors" tree Visible by changing the "+" to "-"

Right Click on "Residuals"



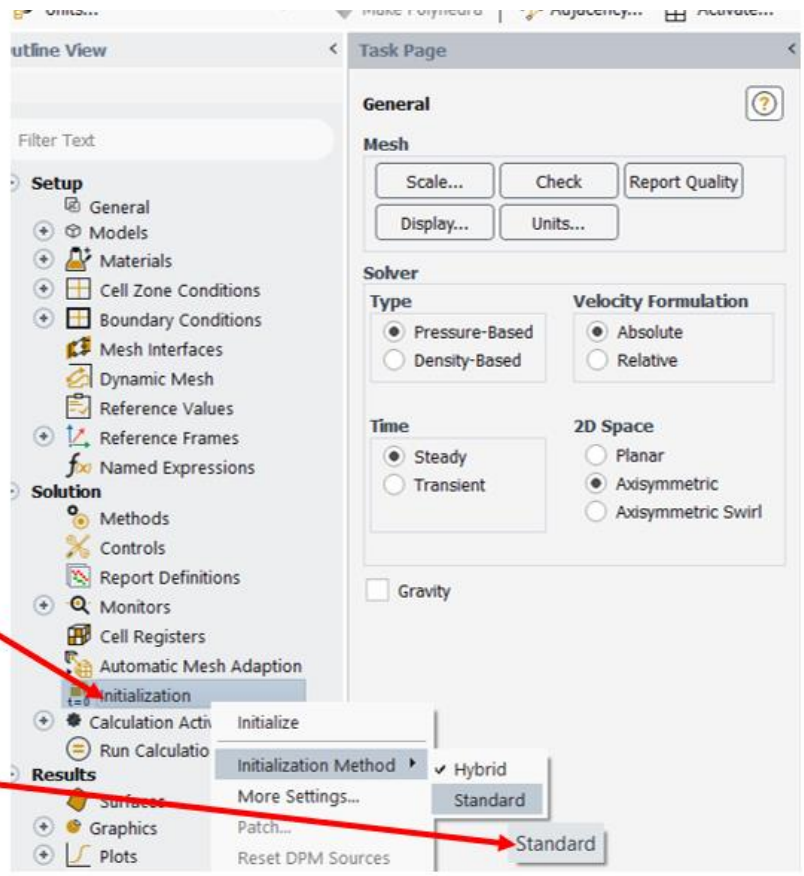

Outline View
Task Page
Filter Text
Setup
General
Models
Materials
Cell Zone Conditions
Boundary Conditions
Mesh Interfaces
Dynamic Mesh
Reference Values
Reference Frames
Named Expressions
Solution
Methods
Controls
Report Definitions
Monitors
Residual
Report Files
Report Plots
Cell Registers
Automatic Mesh Adaption
Initialization
Calculation Activities
Run Calculation
Results
Surfaces
Graphics
Plots
Edit...
Print
Plot
Embed in
General
Mesh
Scale...
Check
Report Quality
Display...
Units...
Solver
Type
Pressure-Based
Density-Based
Velocity Formulation
Absolute
Relative
Time
Steady
Transient
2D Space
Planar
Axisymmetric
Axisymmetric Swirl
Gravity
Click on
“Edit”

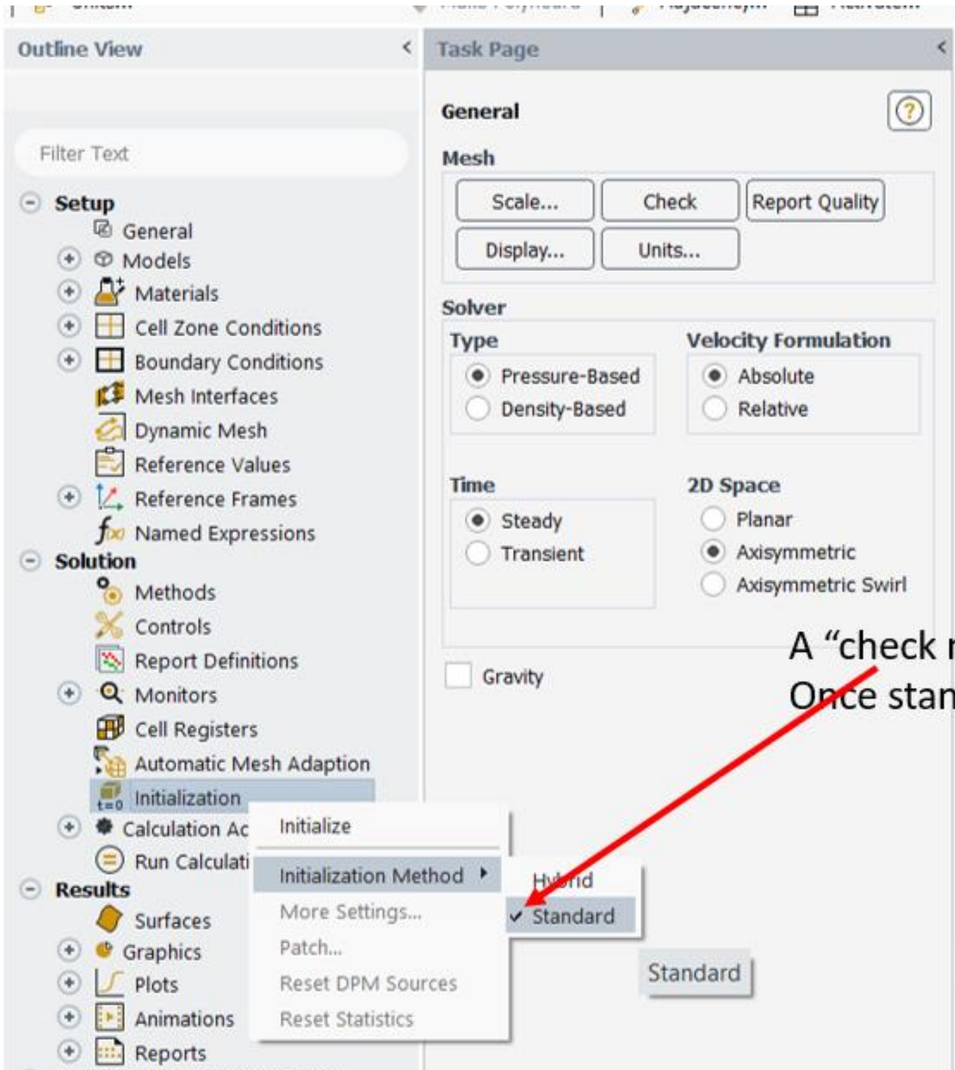


Press “OK” once the numbers are changed.

The iteration process will continue until all equations reach the 1e-6 value. The intent is to increase the solution accuracy.

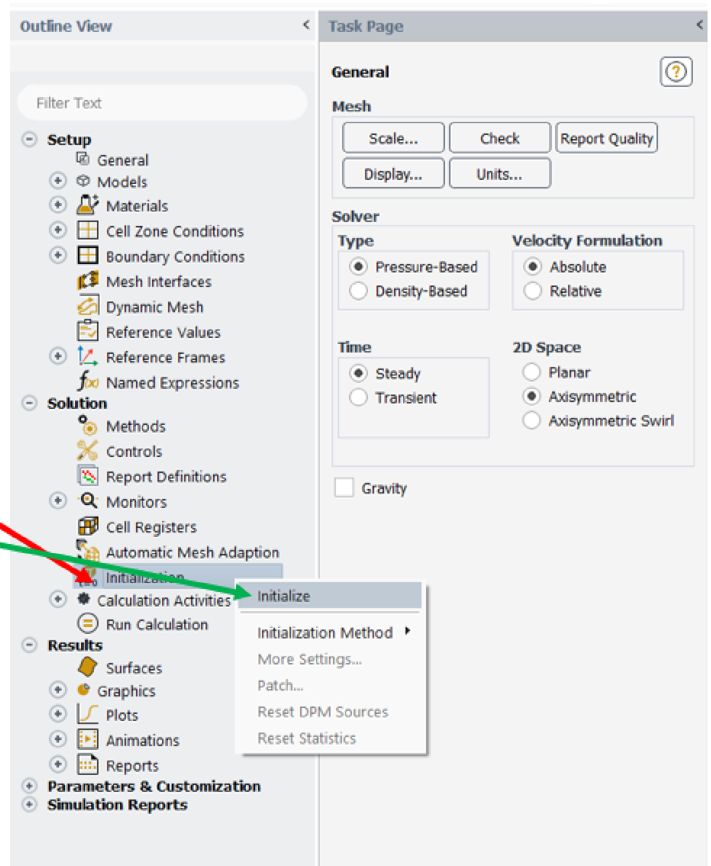
Right click on "Initialization".
Then move the mouse pointer over menu selections until Standard
Is visible.
Then click on "Standard".



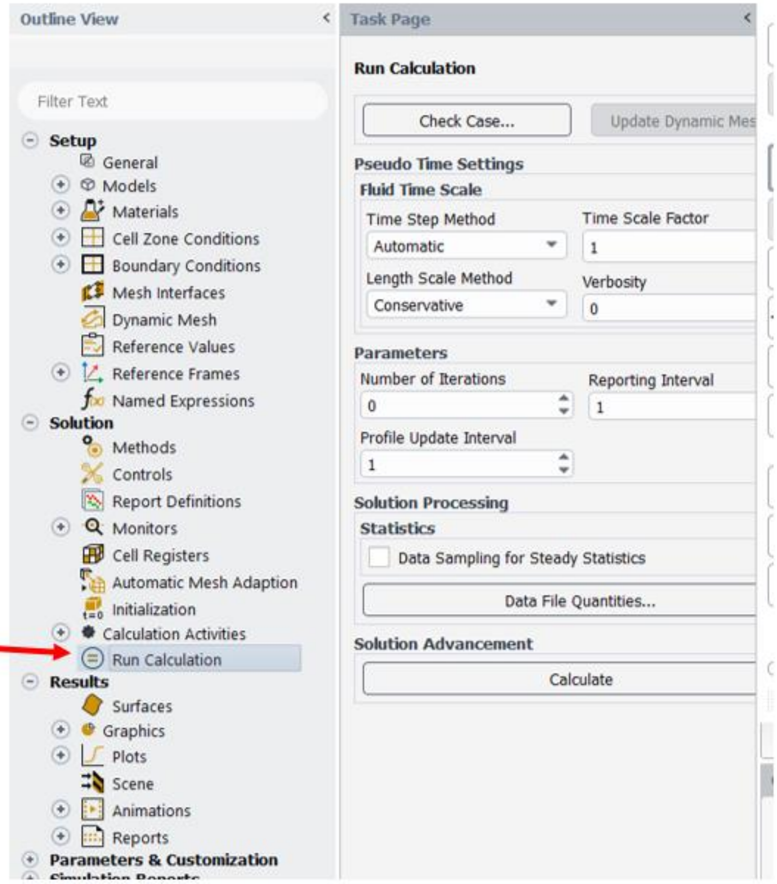


A "check mark" will appear
Once standard is selected.

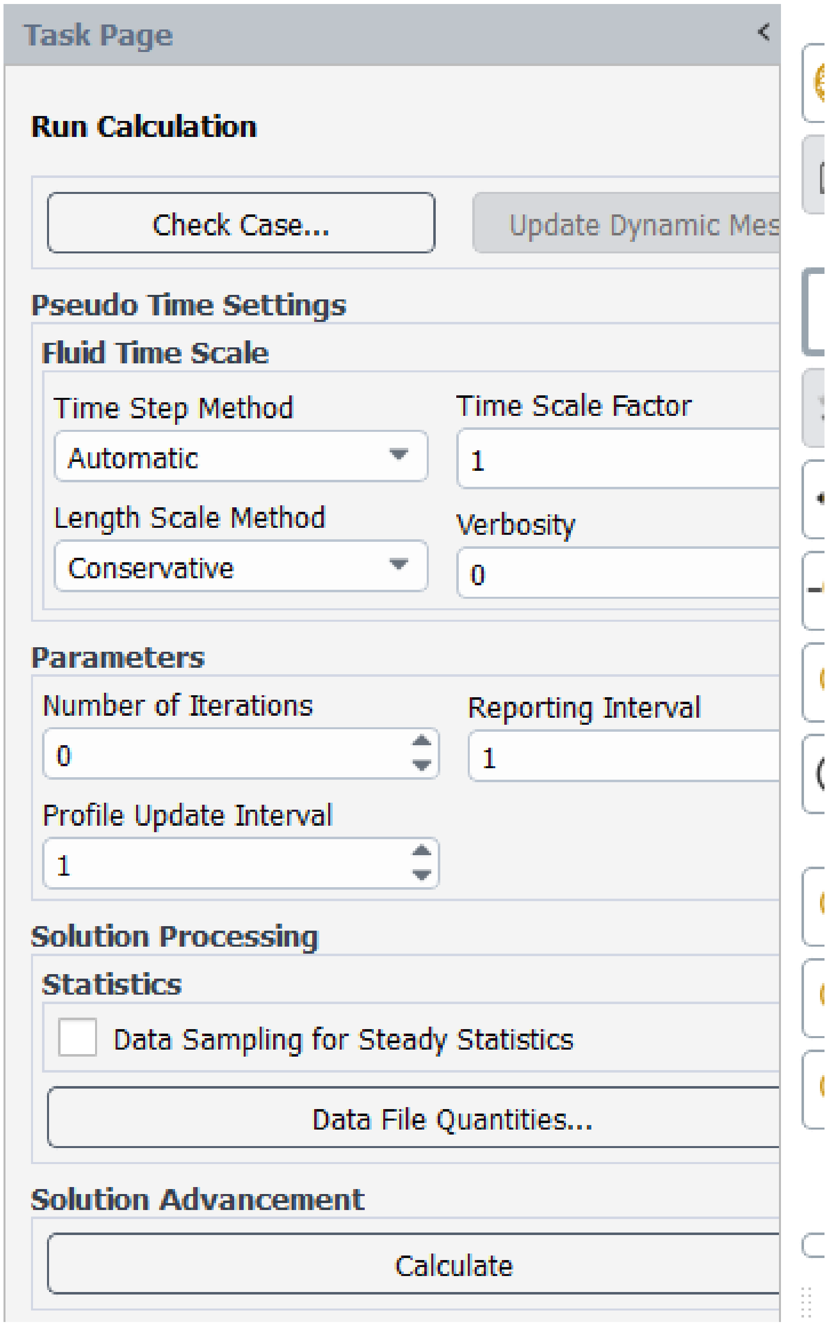
Right Click on
“Initialization”
And then click on
“Initialize”

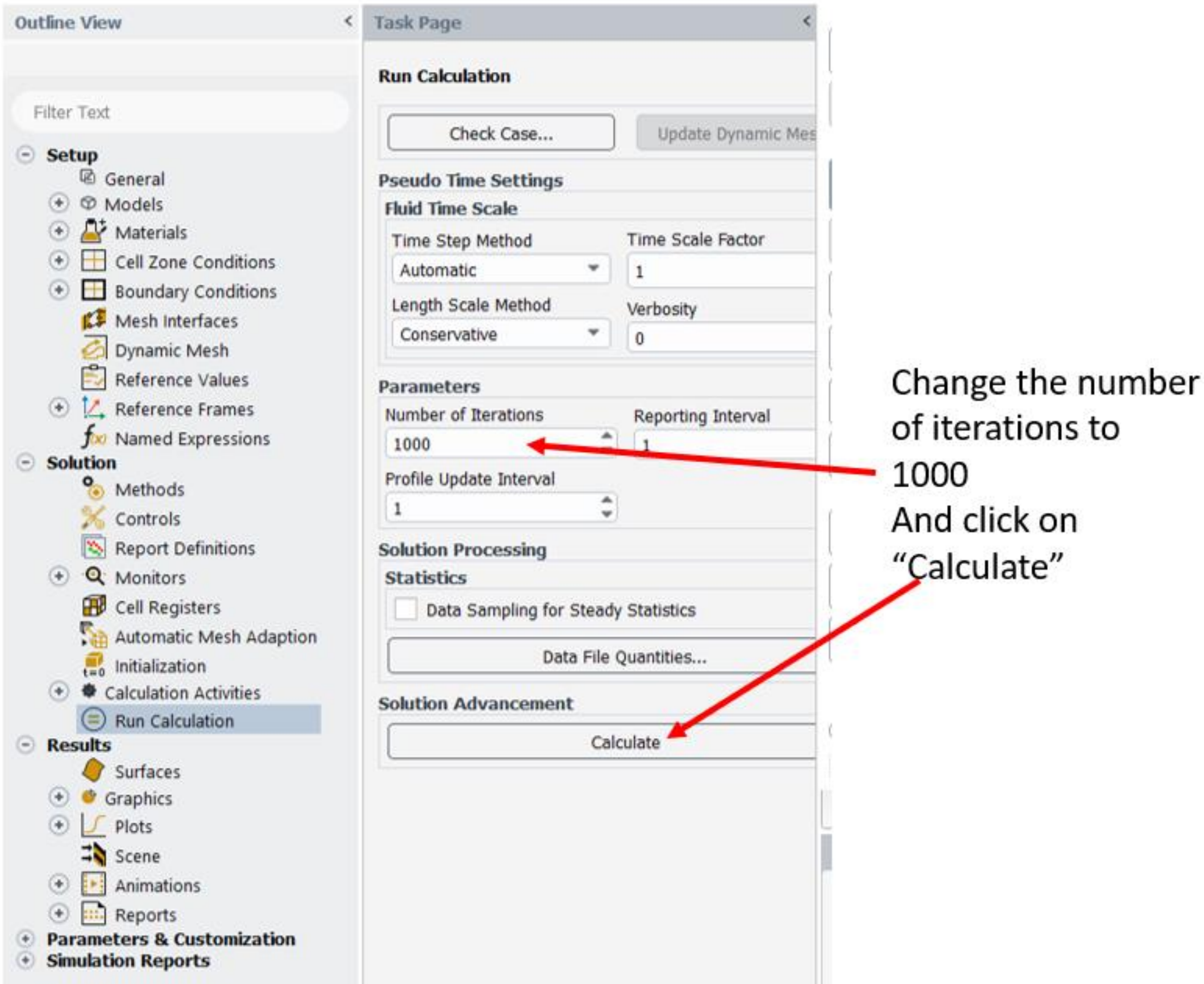


Click on
"Run Calculation"

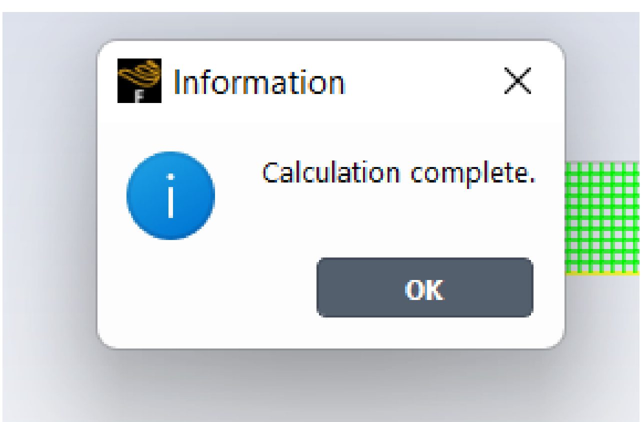


The following dialog box appears as a result of clicking on “Run Calculation”.





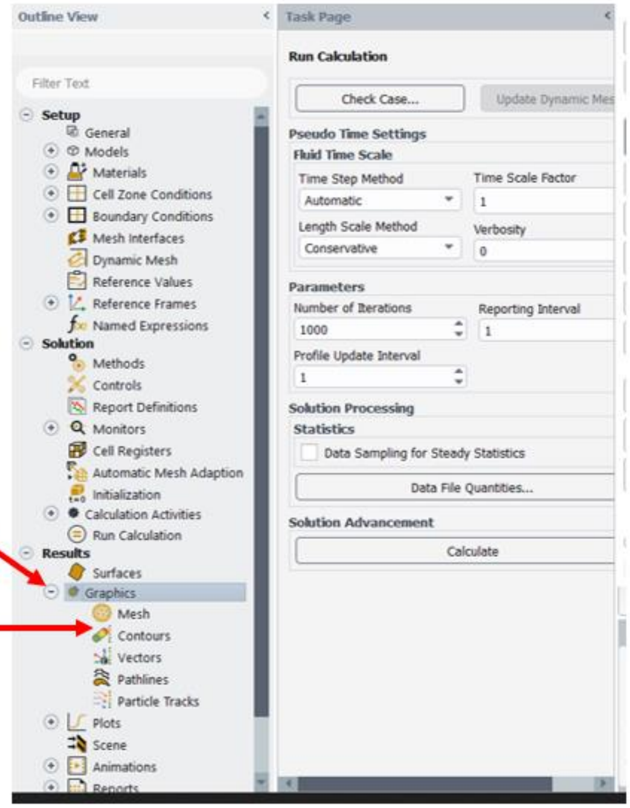
The program runs quickly, and the following message appears.



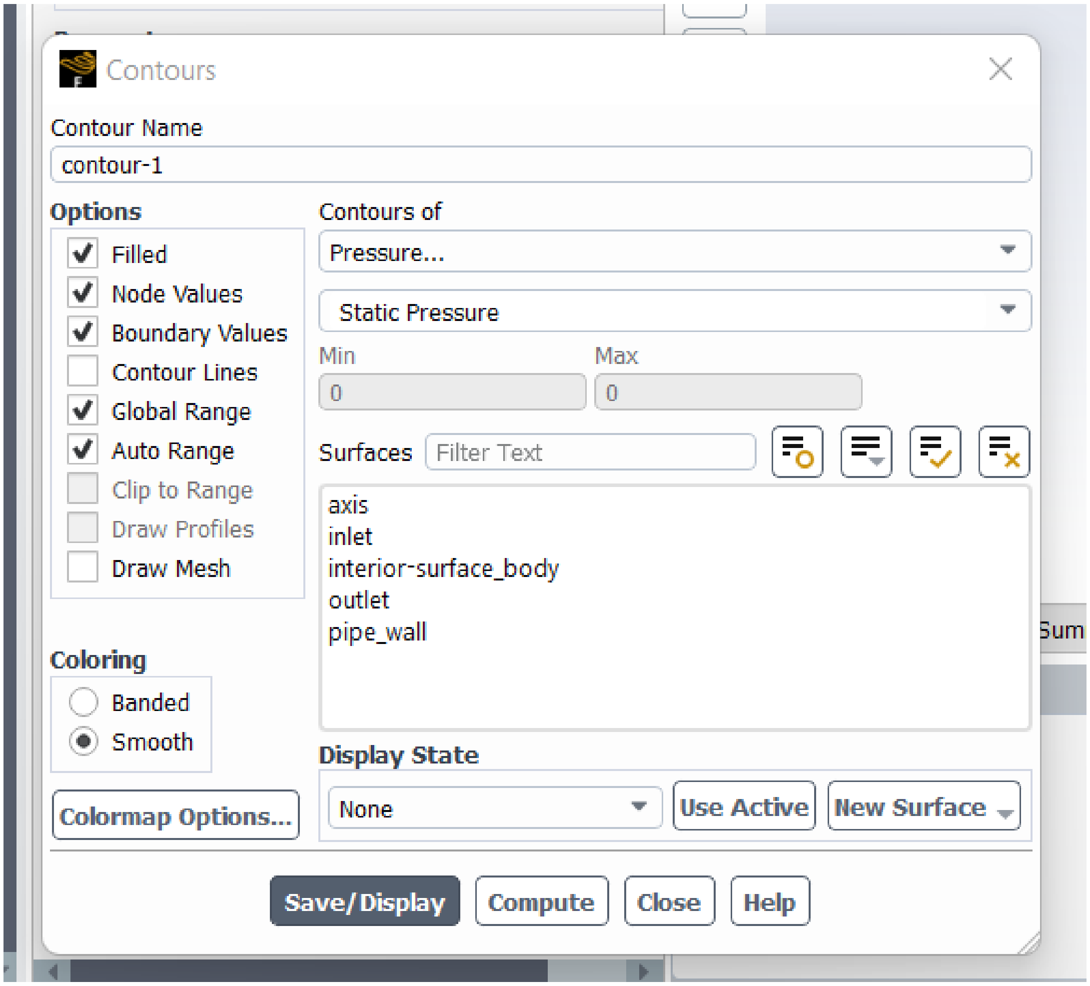
Click on "OK" above, and the dialog box disappears.

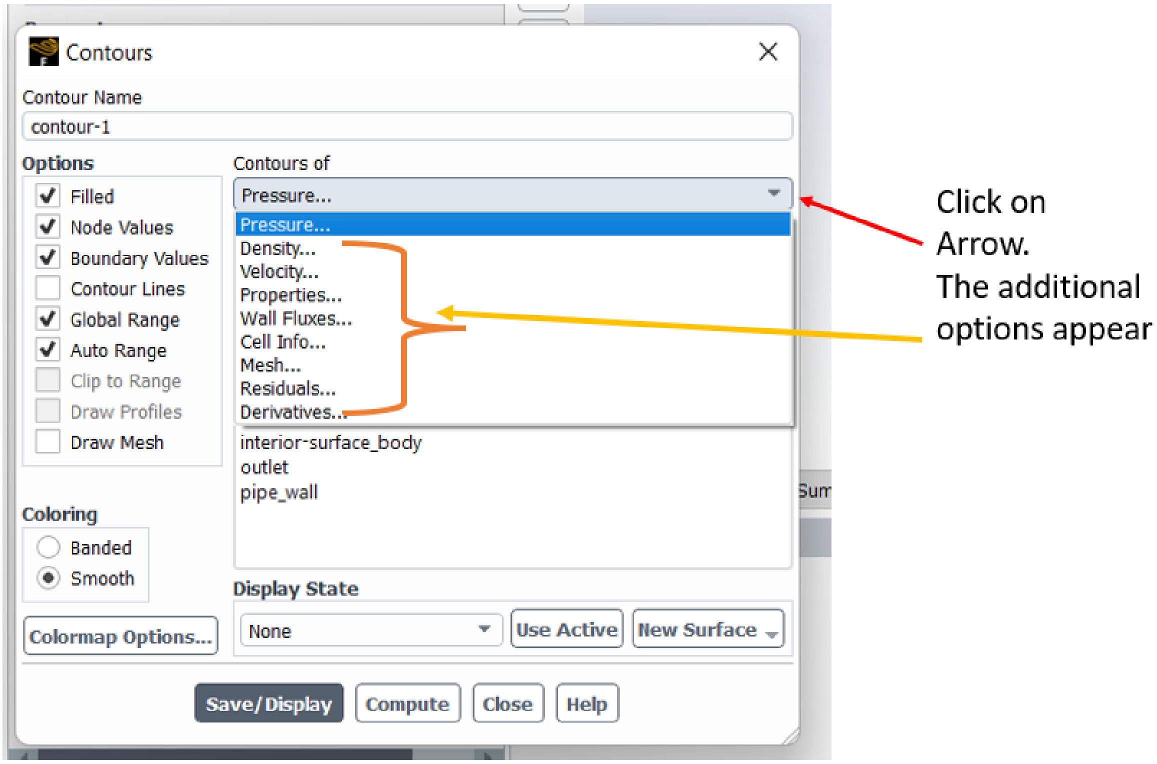
Make the "Graphics" menu appear by changing "+" to "-"

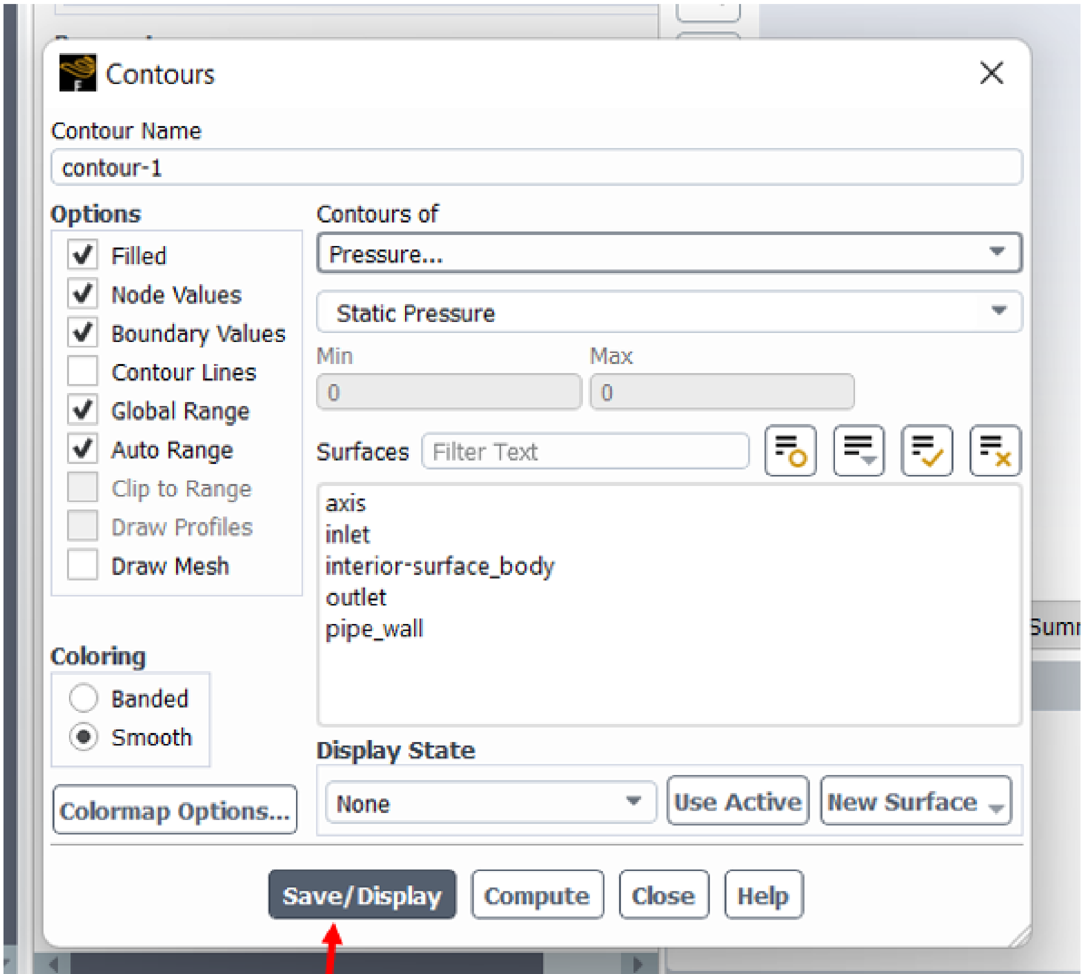
Double Click on "Contours"



The following dialog box appears.

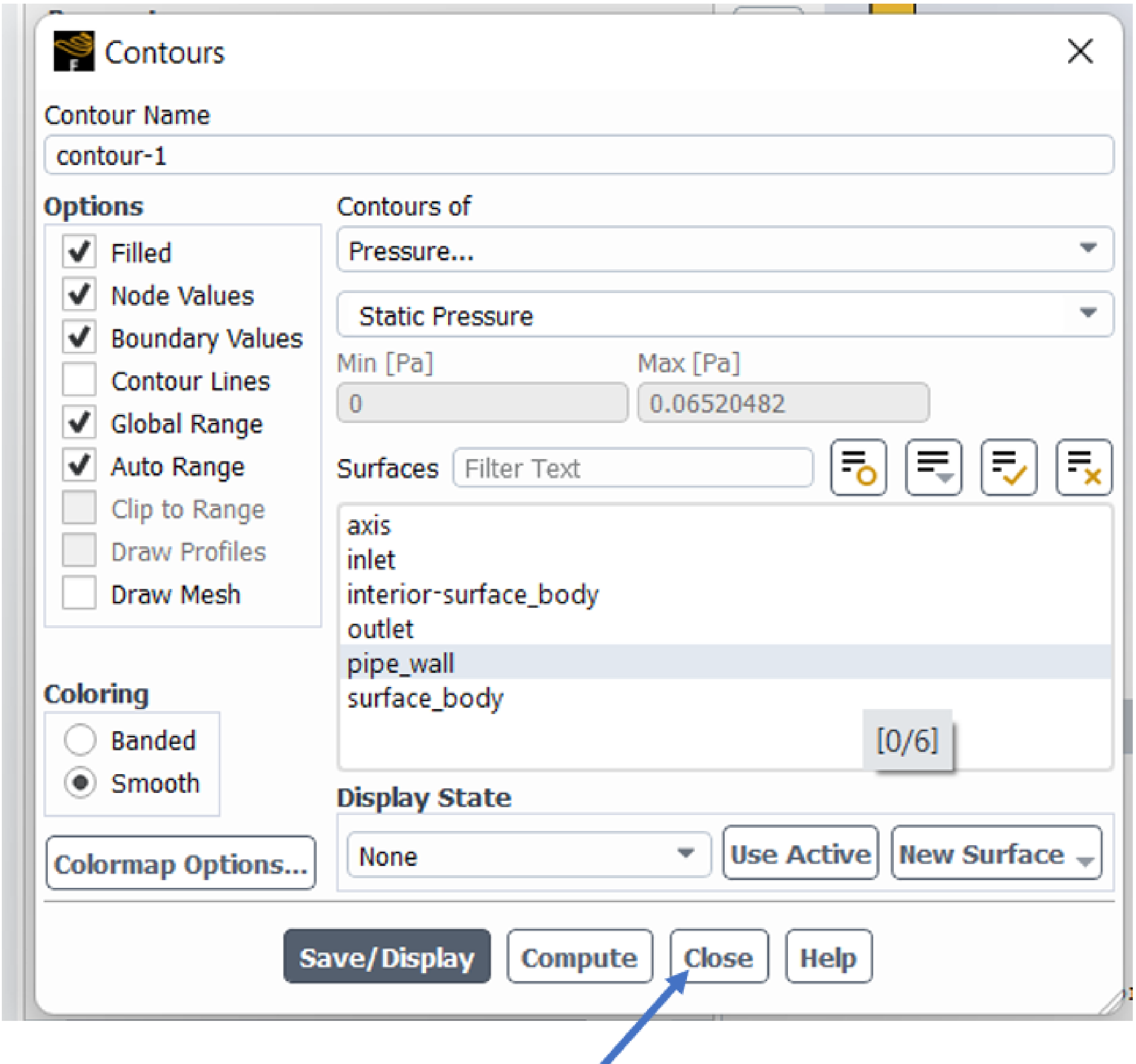



Contours
Contour Name
contour-1
Options
Filled
Node Values
Boundary Values
Contour Lines
Global Range
Auto Range
Clip to Range
Draw Profiles
Draw Mesh
Contours of
Pressure...
Pressure...
Density...
Velocity...
Properties...
Wall Fluxes...
Cell Info...
Mesh...
Residuals...
Derivatives...
interior-surface_body
outlet
pipe_wall
Coloring
Banded
Smooth
Display State
None
Use Active
New Surface
Colormap Options...
Save/Display
Compute
Close
Help
Click on Arrow.
The additional options appear



Click on
'Save/Display"

The following is the pressure contour.

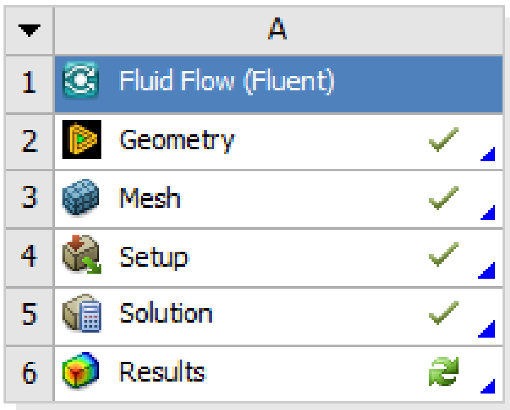


Click on "Close" to
make box invisible

At this point save the project through the WB window. By doing so, all open windows in addition to the WB window close.

The contour plots can be saved and deleted. The details are shown below.

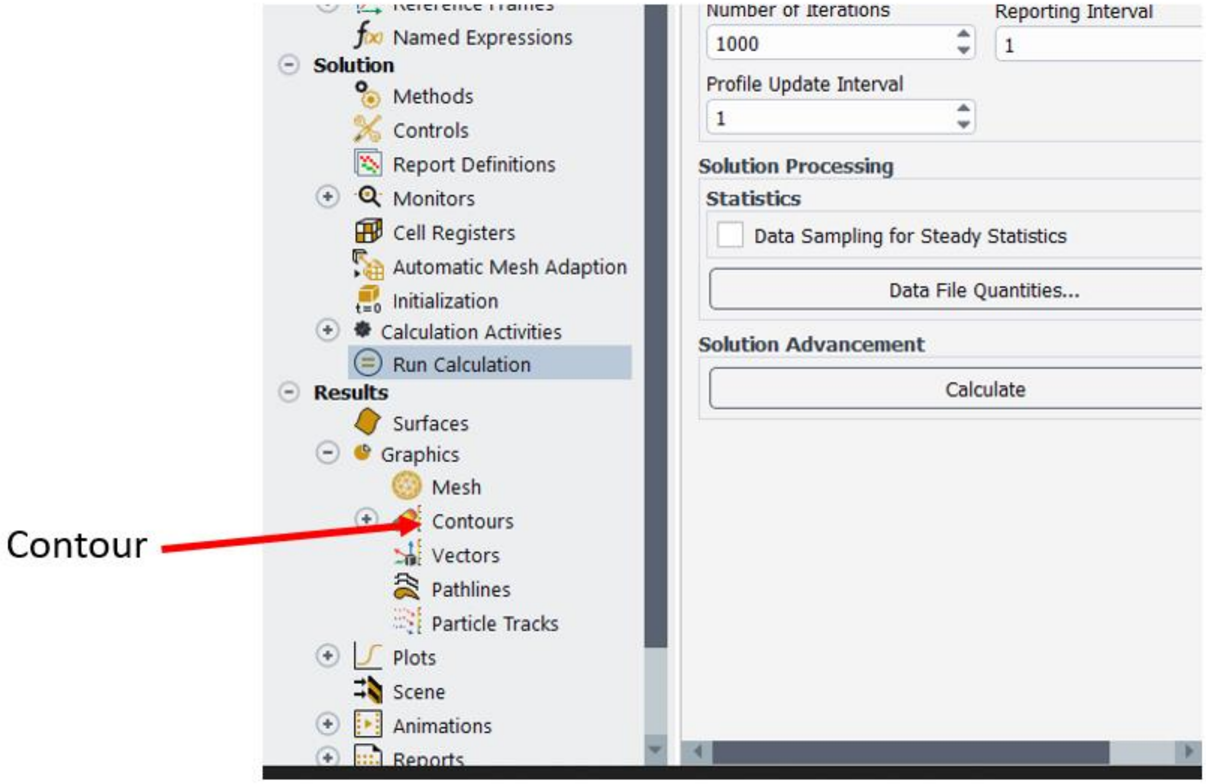
After the project is recalled, the screen appears as shown below. Since the project was carried through the solution step, the "Geometry", "Mesh", "Setup" and "Solution" steps have a checkmark next to them. However, until and unless each element is double clicked, the screen associated with that choice is not available at the bottom of the windows screen.



Laminar fluid flow

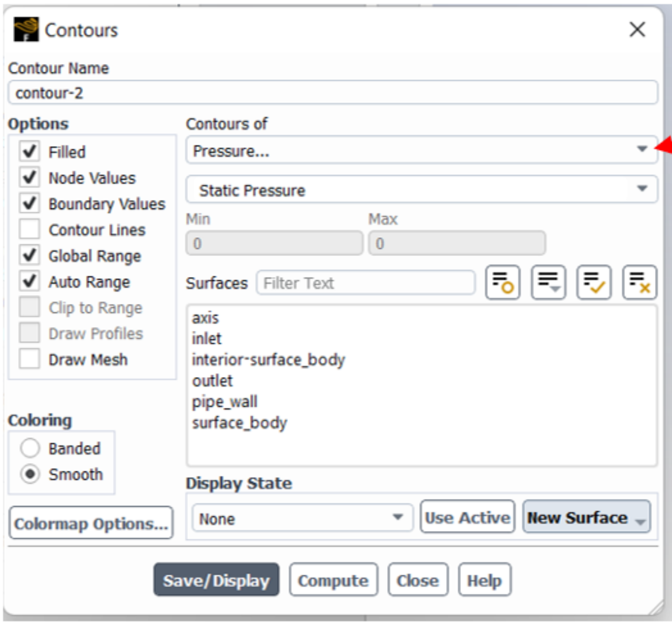
Double click on “Solution” on WB screen to bring up the “f” screen (if it is not already available).

On the tree on the left of the “f” screen, “Contours” will be used.

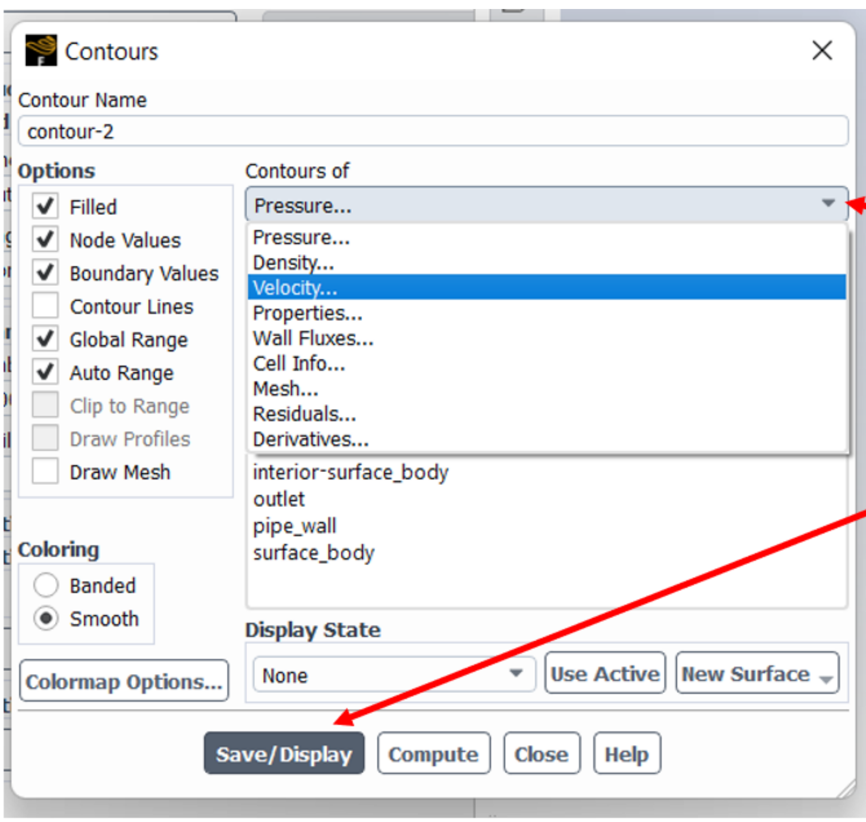


Double click on “Contour”.

The following appears as a result of double clicking on “Contour”.

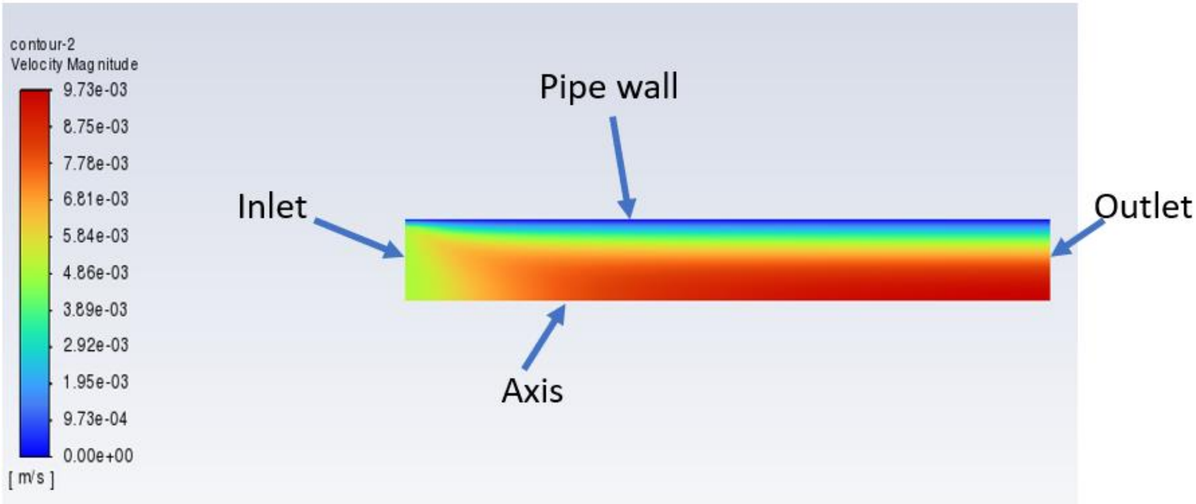


By clicking on this arrow different parameters that can be displayed become visible.

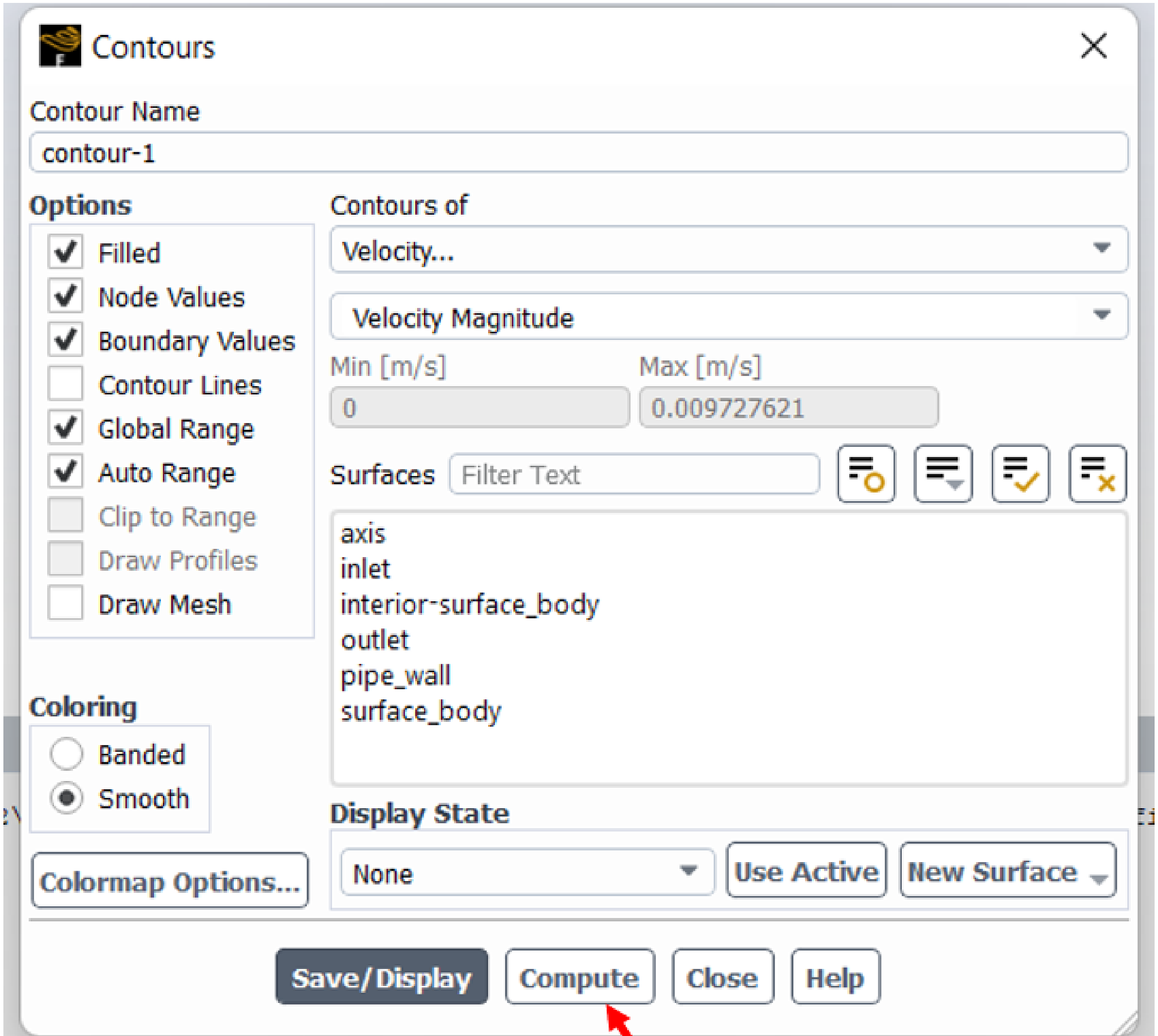


Click on Arrow.
Click on “Velocity...”
Click on “Save/Display”

The following velocity profile is displayed and saved. Note that the velocity profile is for half the pipe and not the entire pipe.

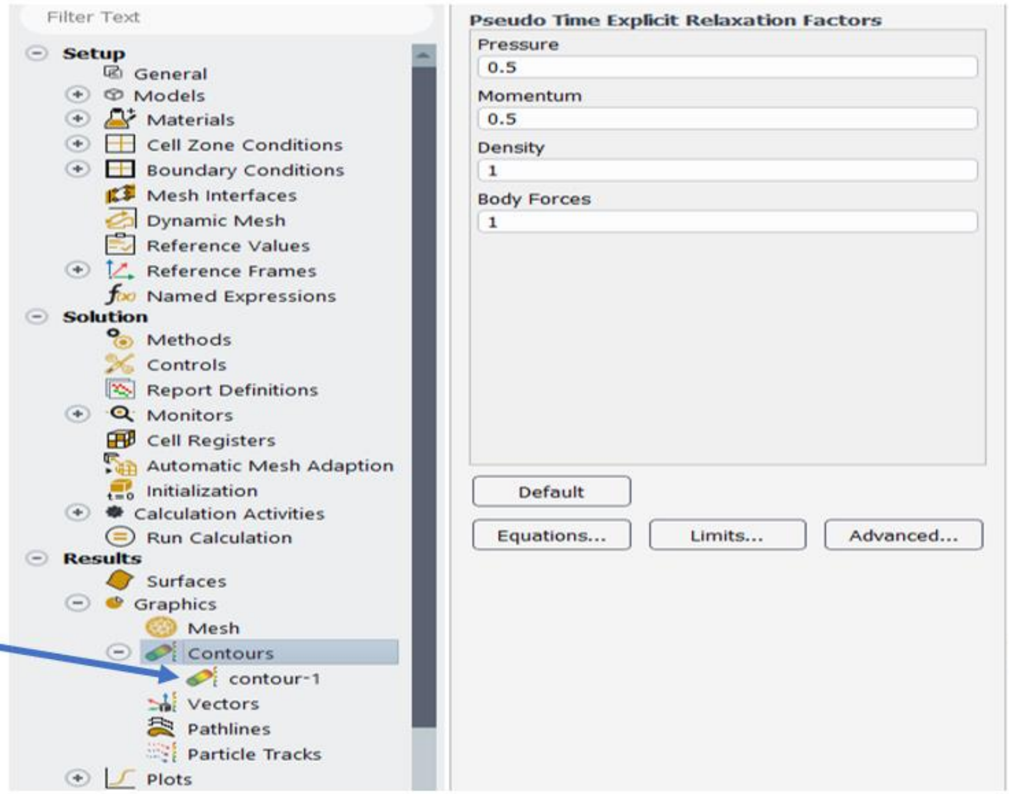


The model appears as shown above because the "Axisymmetric" option was chosen earlier.

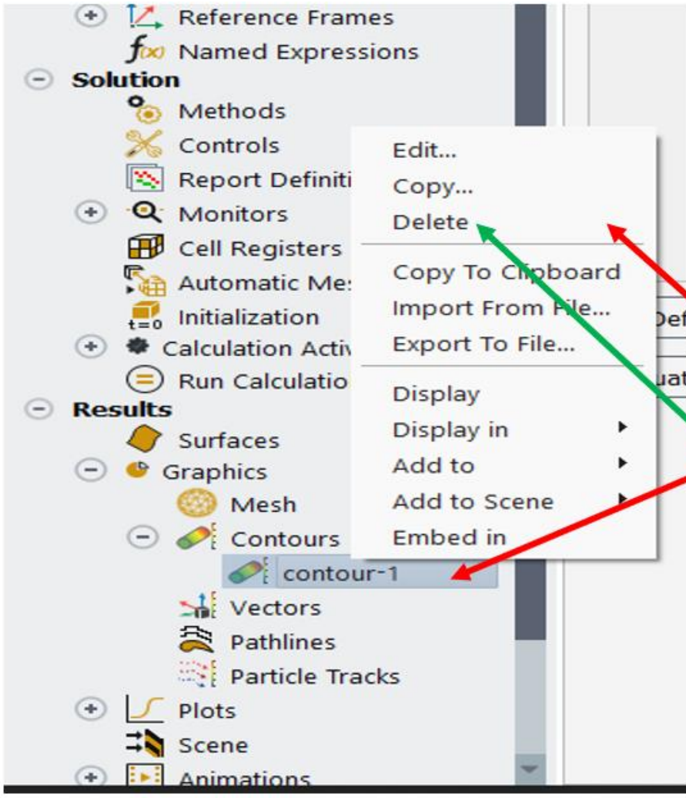


Click on “Close” to make this box disappear

The tree on the left shows that the contour is saved.



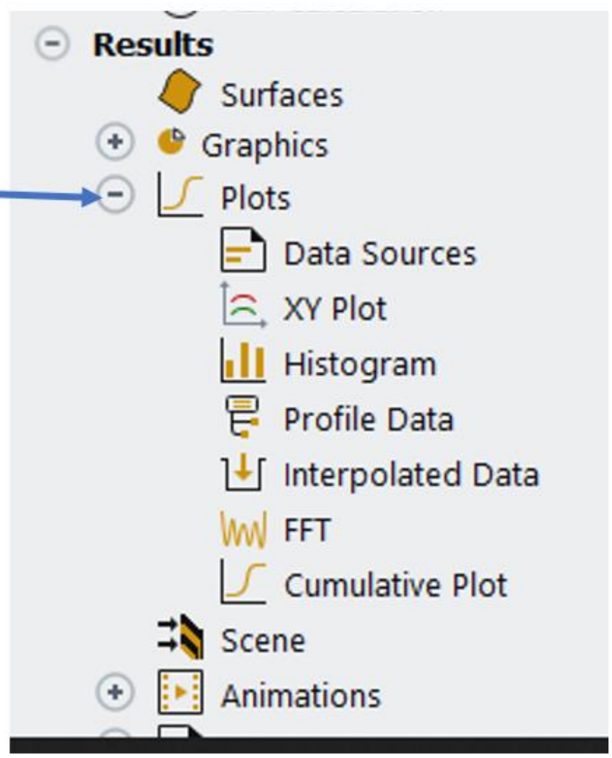
"Contour 1" is the velocity profile just saved.



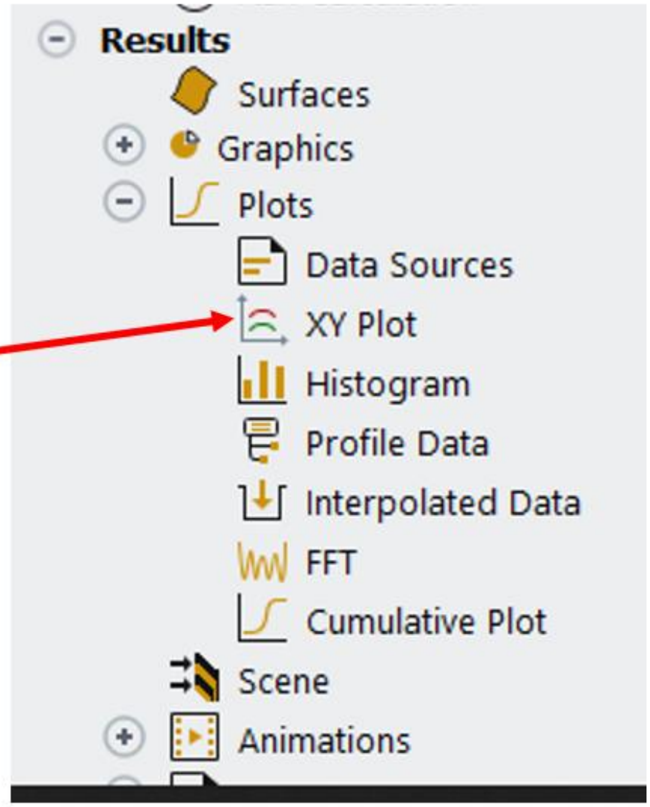
Right click on “contour 1” and this box appears, and the contour can be deleted by clicking here

Make the plot tree available on the left of “f” window.

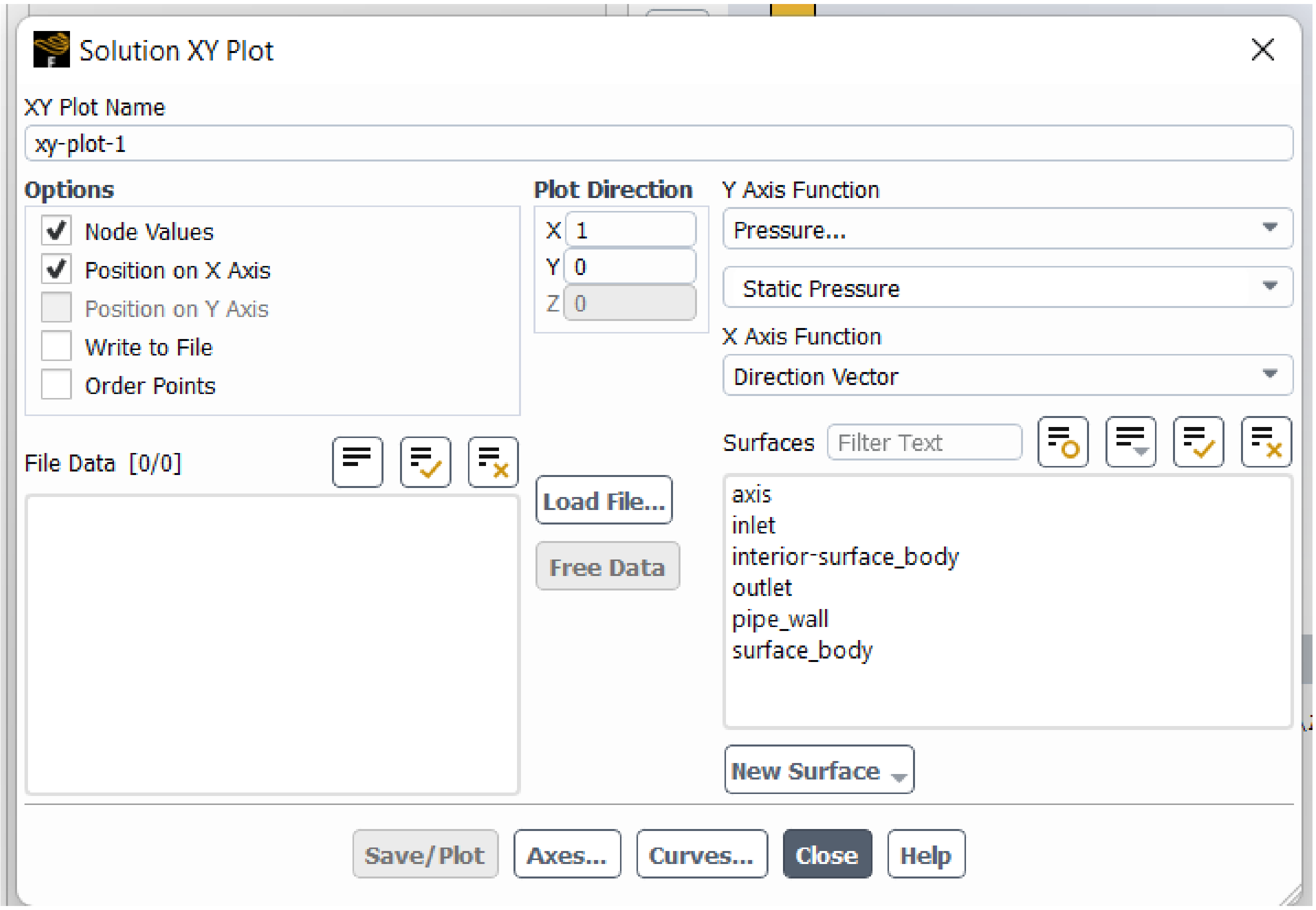
Click on this when it is “+” to make the plots choices visible as shown here.

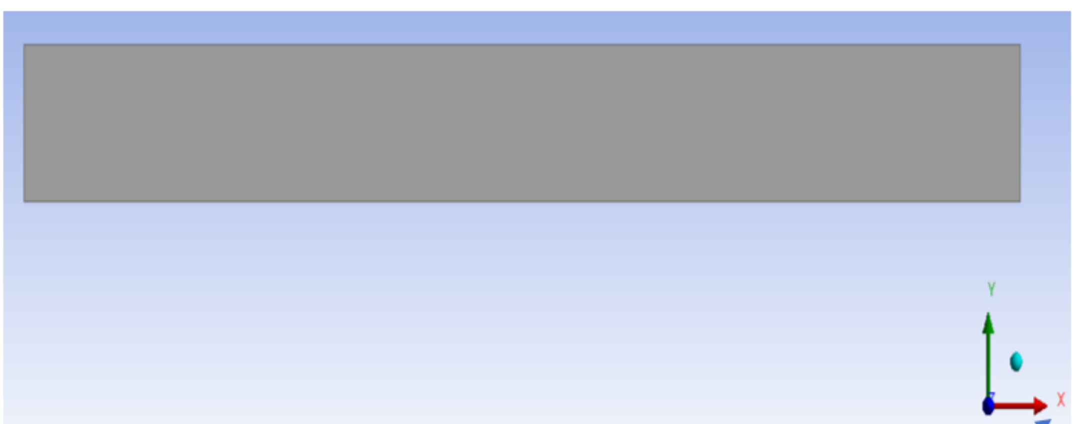


Double click on
"XY Plot"



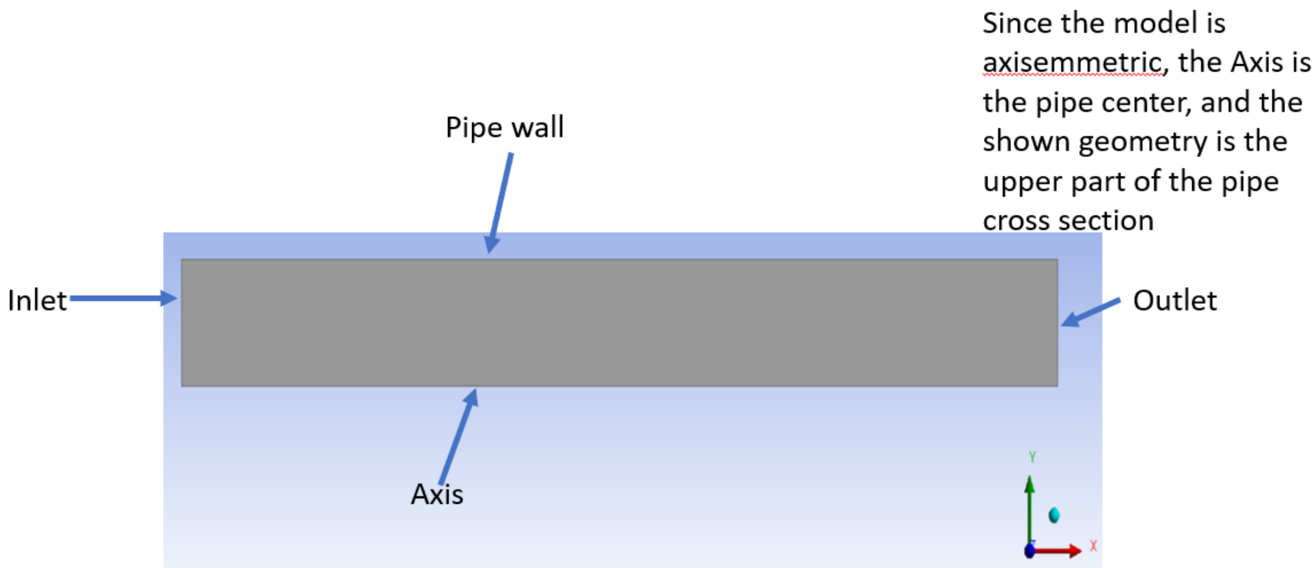
After choosing XY “Plot” the following appears.

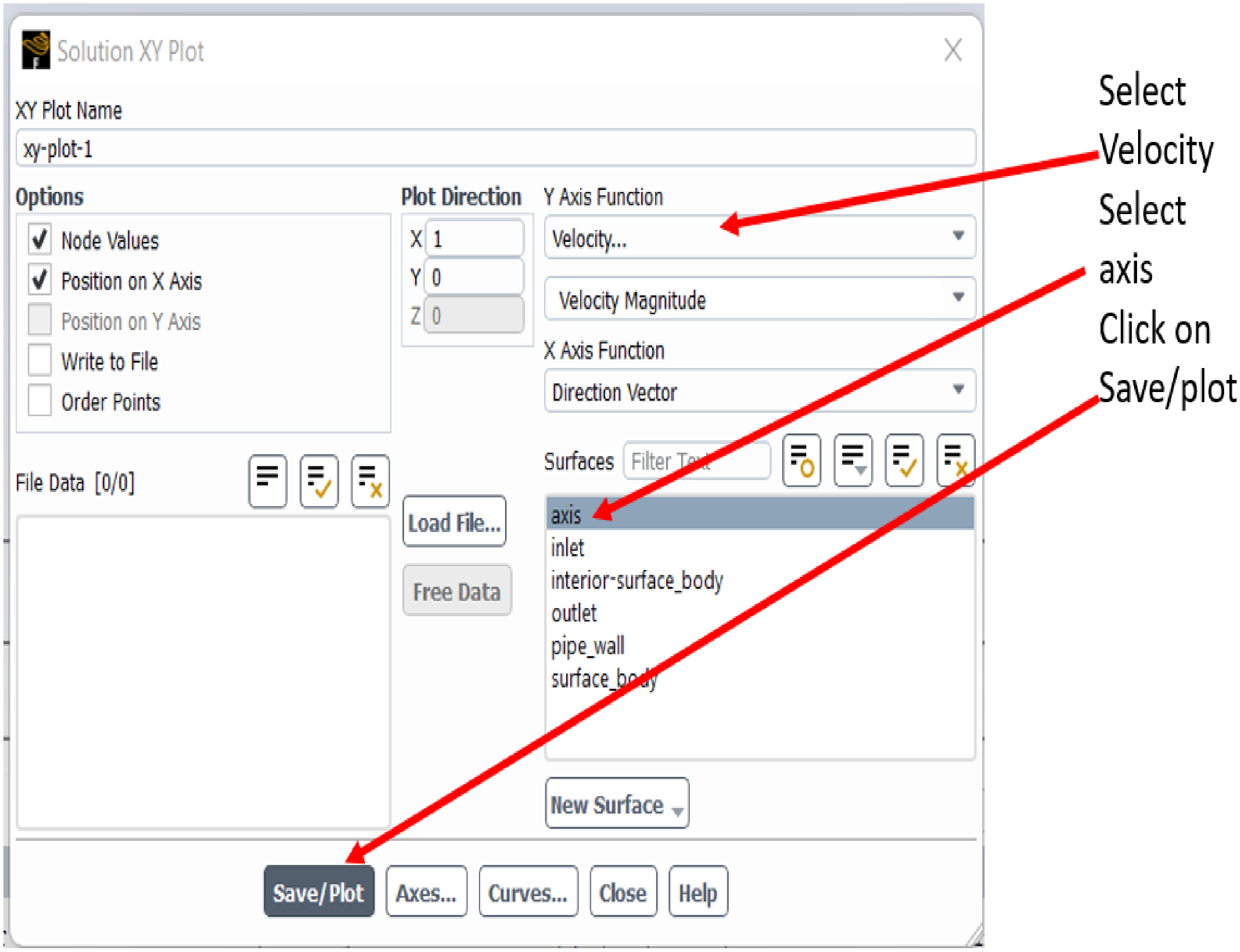




Recall that the pipe was laid in XY plane, in X direction in a 2D fluid model. Consequently, the desired velocity profile is in X direction.

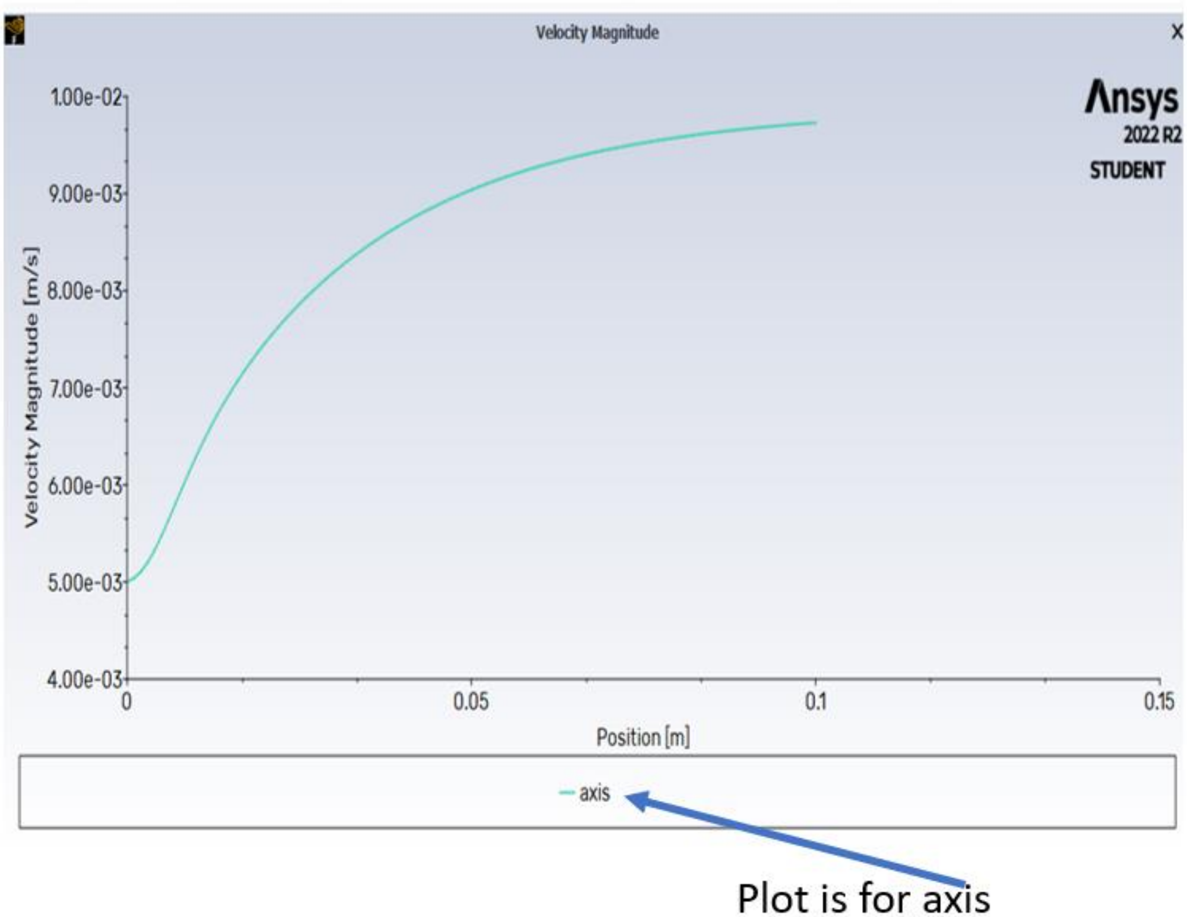
Recall that the following names were assigned to the sides of the pipe.



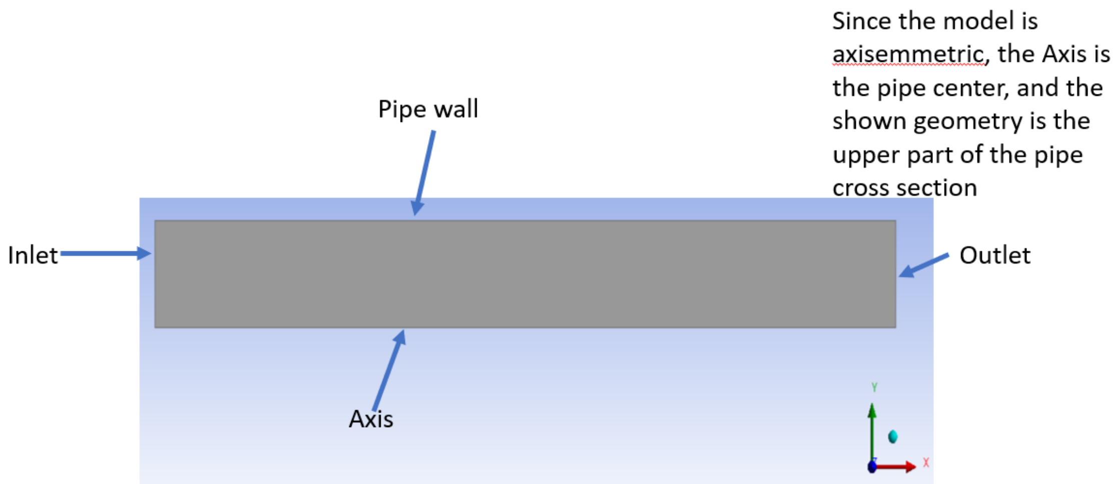


Click on “Close” to close the above box.

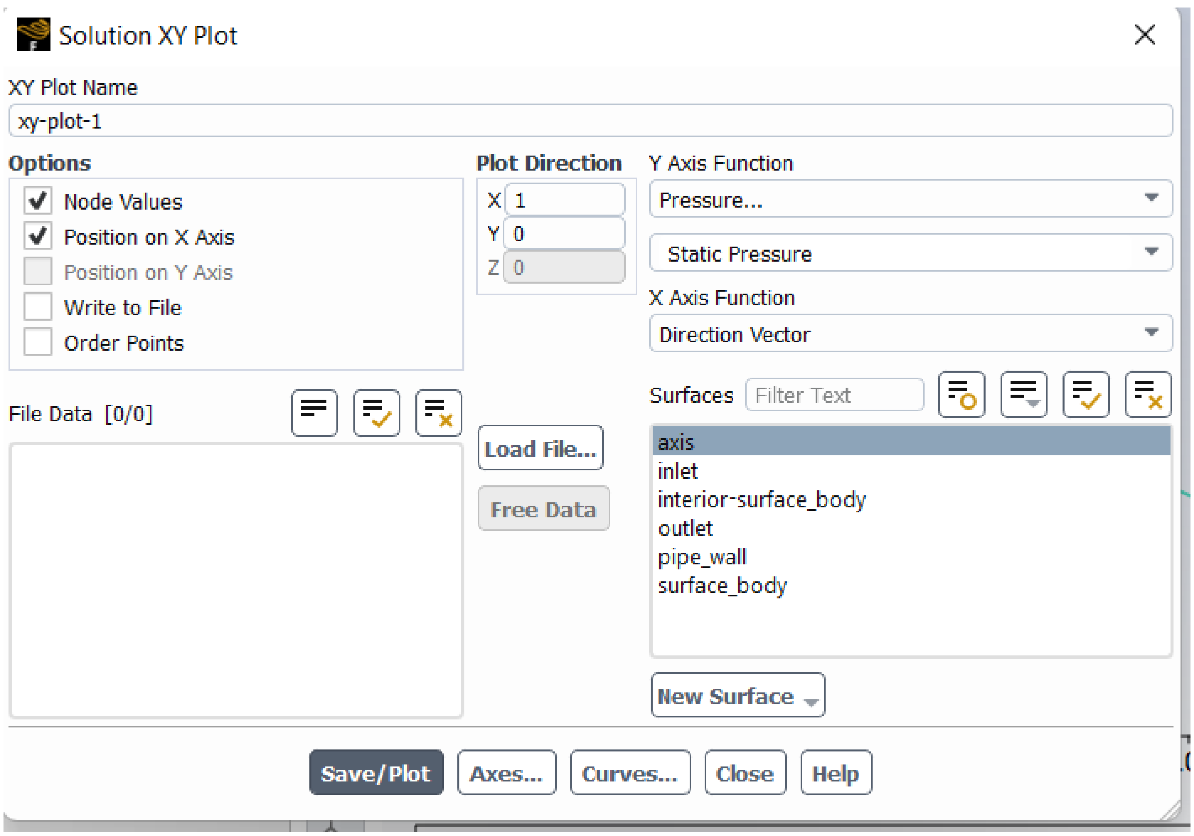
The following plot is generated.



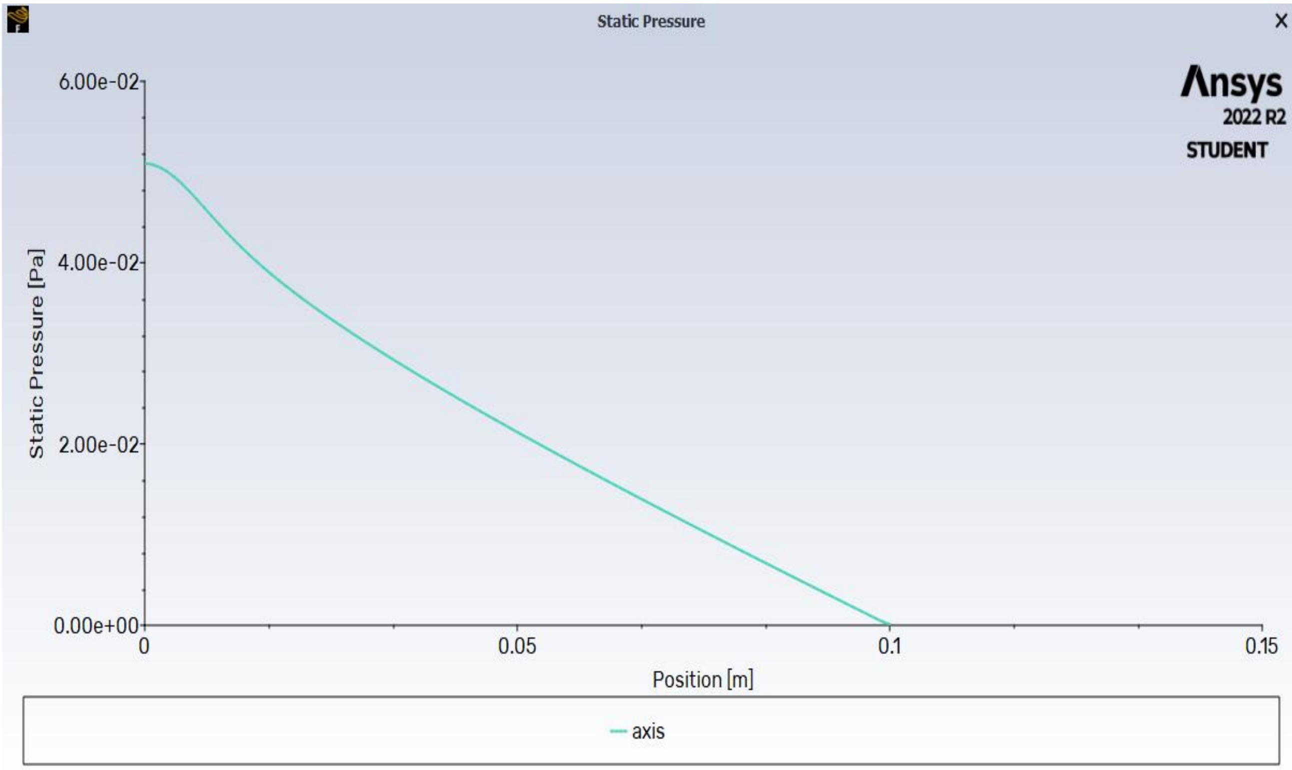
The plot is showing that the velocity at X position “0” (Inlet) is 0.005 m/s and at X position 0.1 m (100 mm which is pipe length) is 0.01 m/s. The plot is for Axis, which is shown below. Axis is the pipe center since the model is axisymmetric.

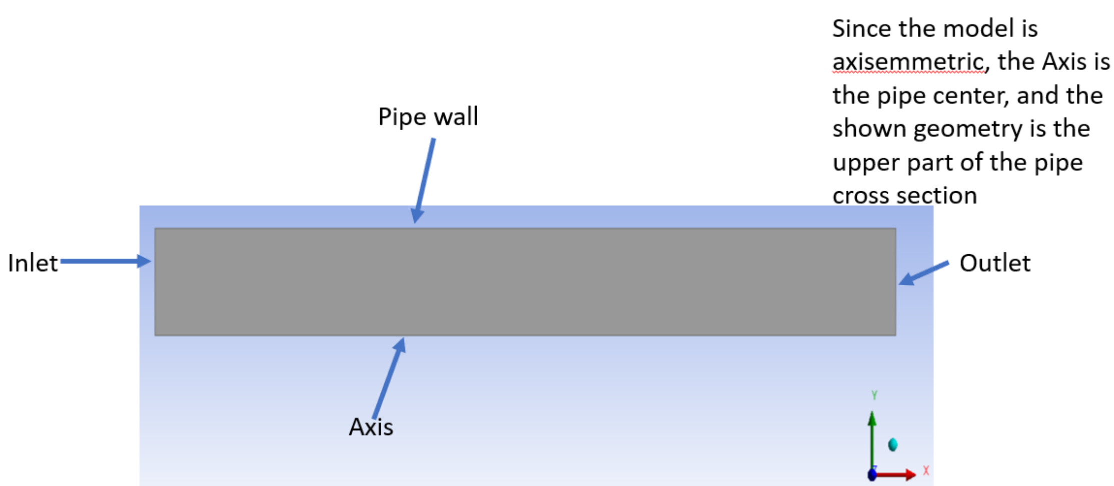


The pressure along the pipe length can be plotted by the following choices. Specifically, notice the choice of pressure.

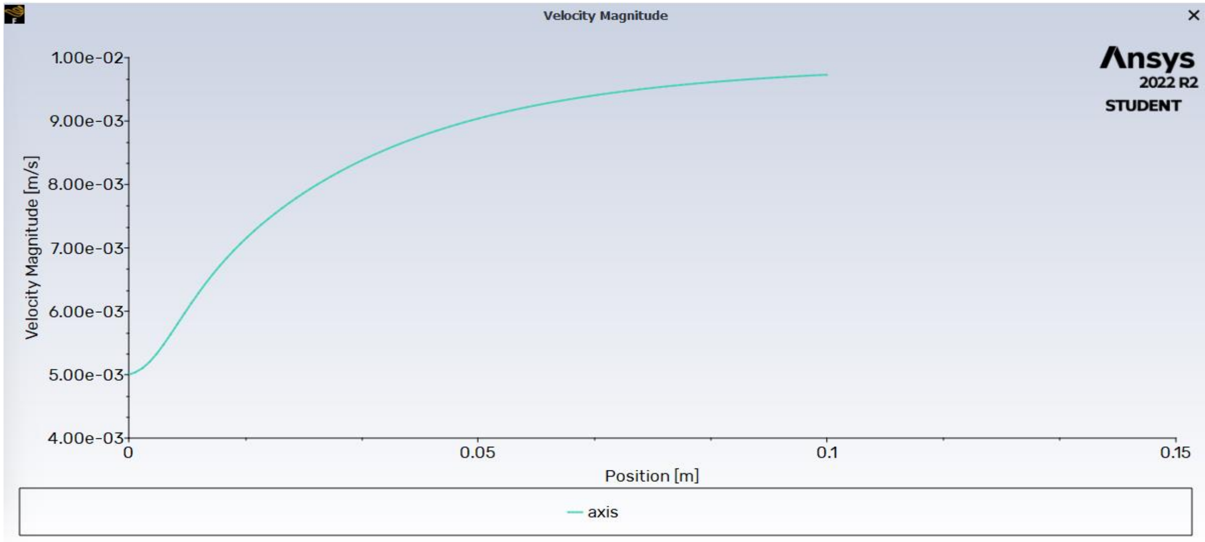


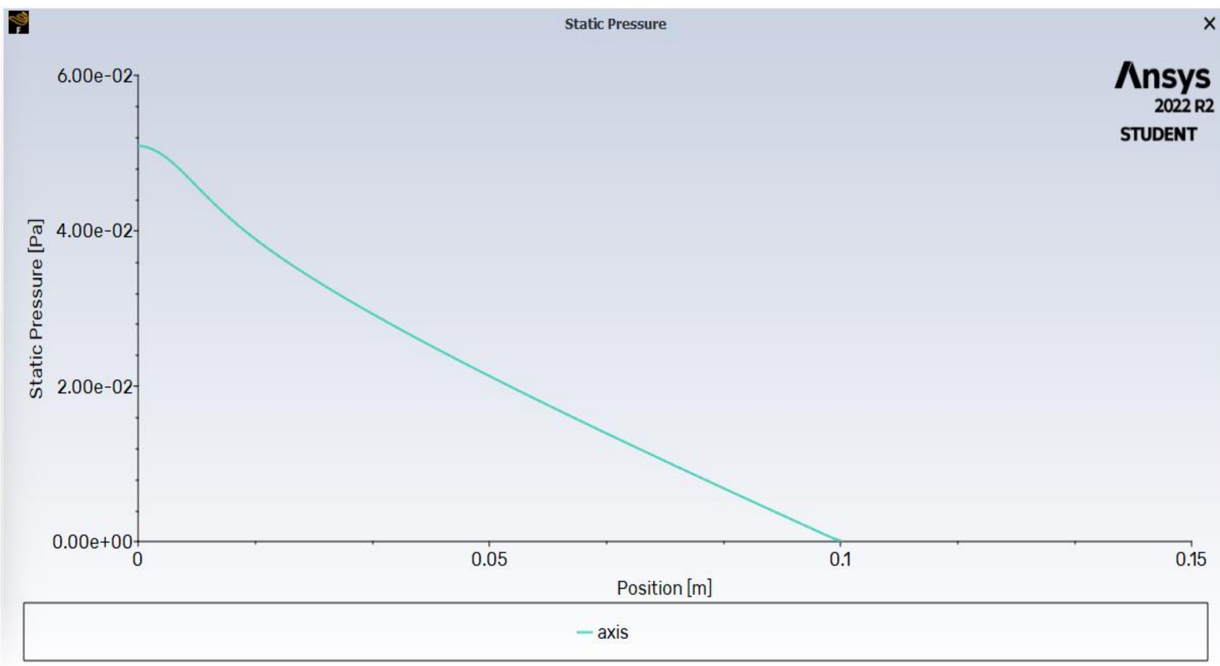
The following is showing the pressure plot along the pipe length.

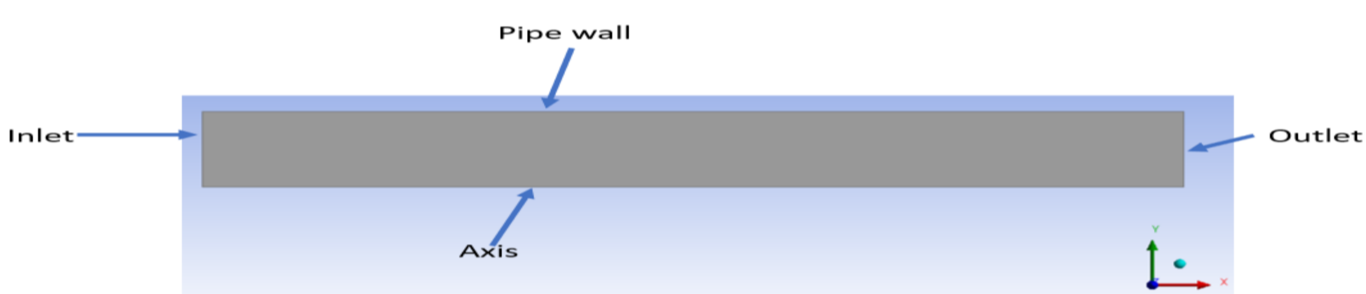




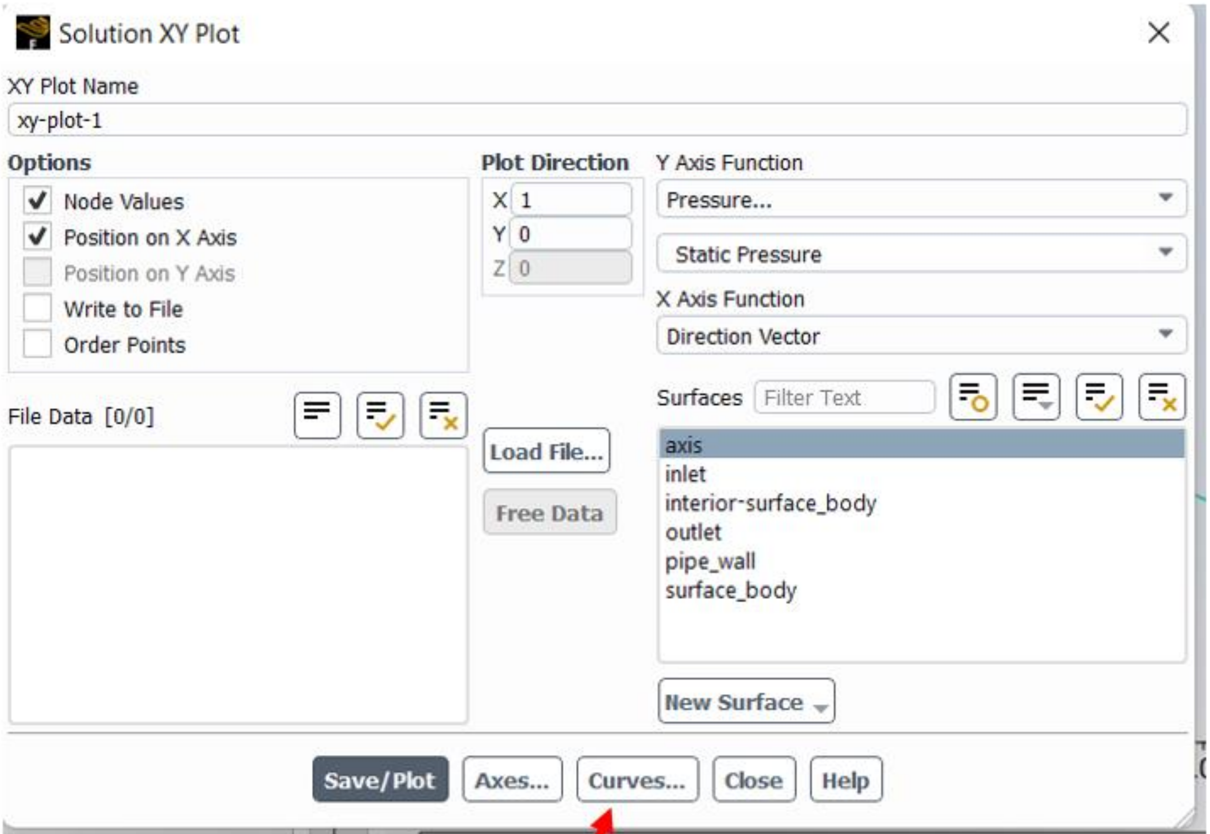
The following shows the velocity and pressure along the pipe length.







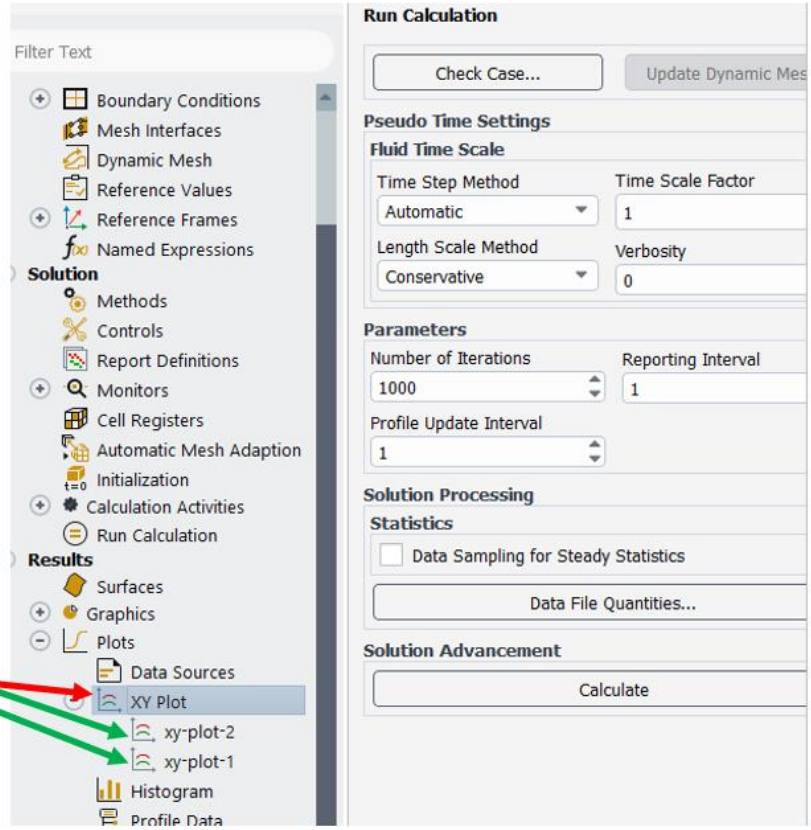
Position in both XY plots is along the axis from left to right. The simulation is showing that as the pressure decreases along the length of the pipe, the velocity increases.



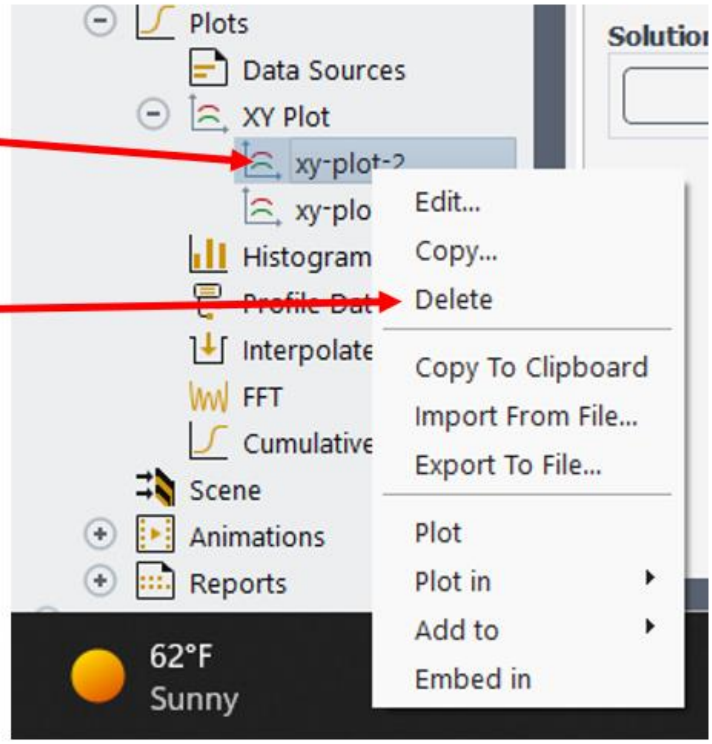
The plot appearance can be changed by clicking on "Curves…", and changing the options

The plots generated are available on the tree on the left side of the “f” screen as shown below.

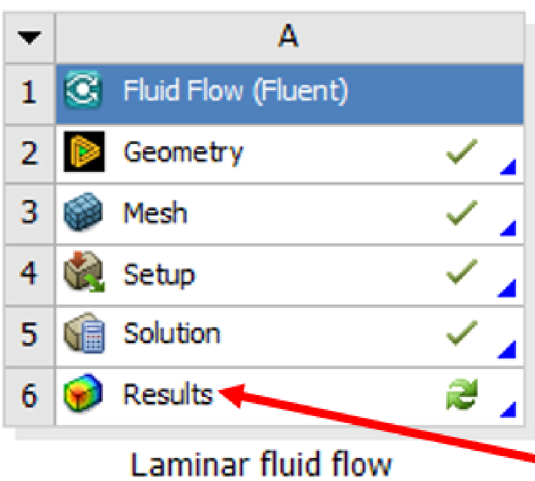
XY plots for velocity and pressure are available under XY plot



Right click on a plot name,
And the box appears that gives choices including the
"Delete"
option.

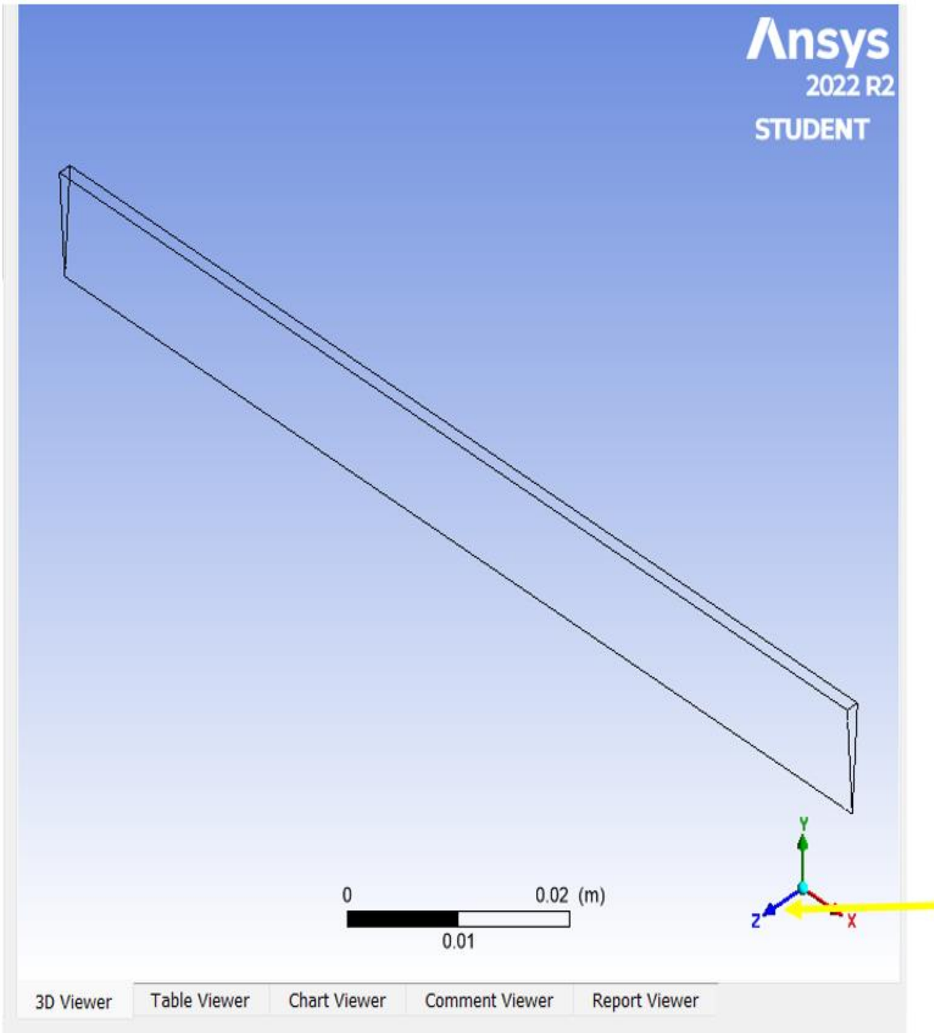


At this point, save the project though the "WB" screen.



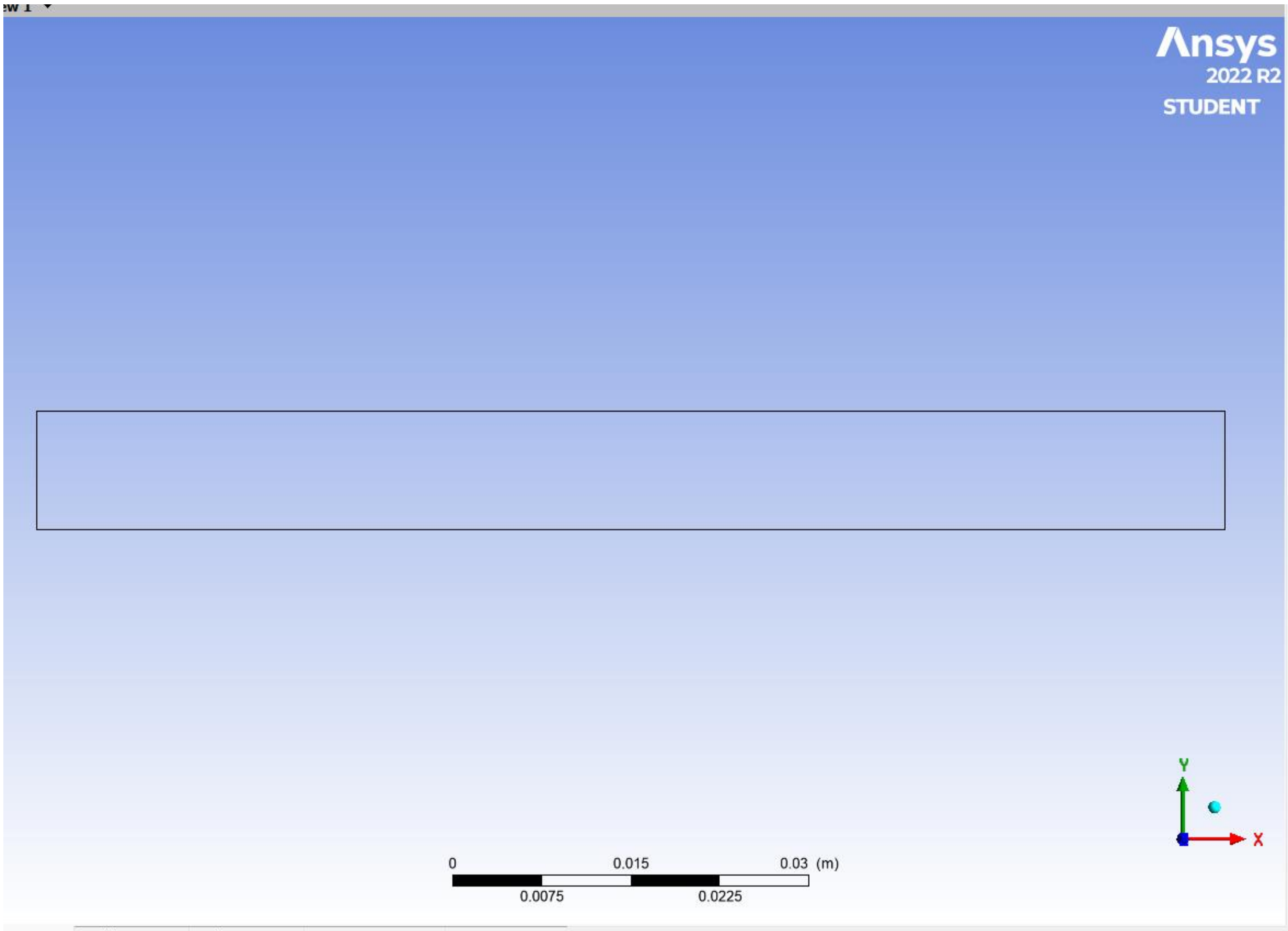
Double click on
"Results"

The following screen identified as “PST” which stands for “CFD-Post” appears.



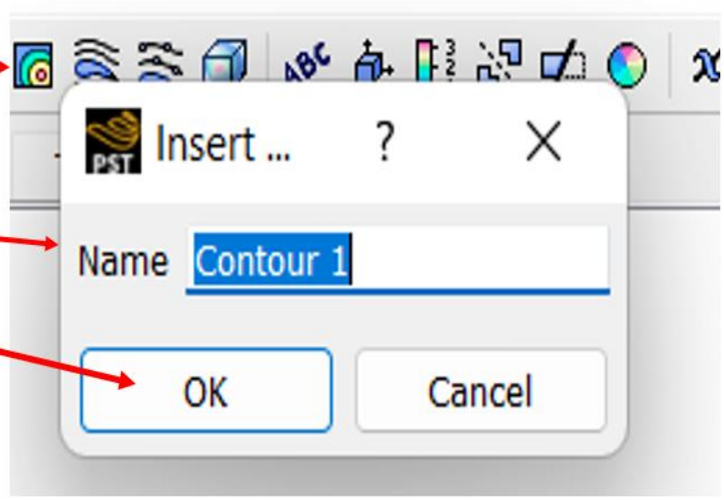
Click on
“Z” to change the view to
XY plane.

The XY view appears as shown below.

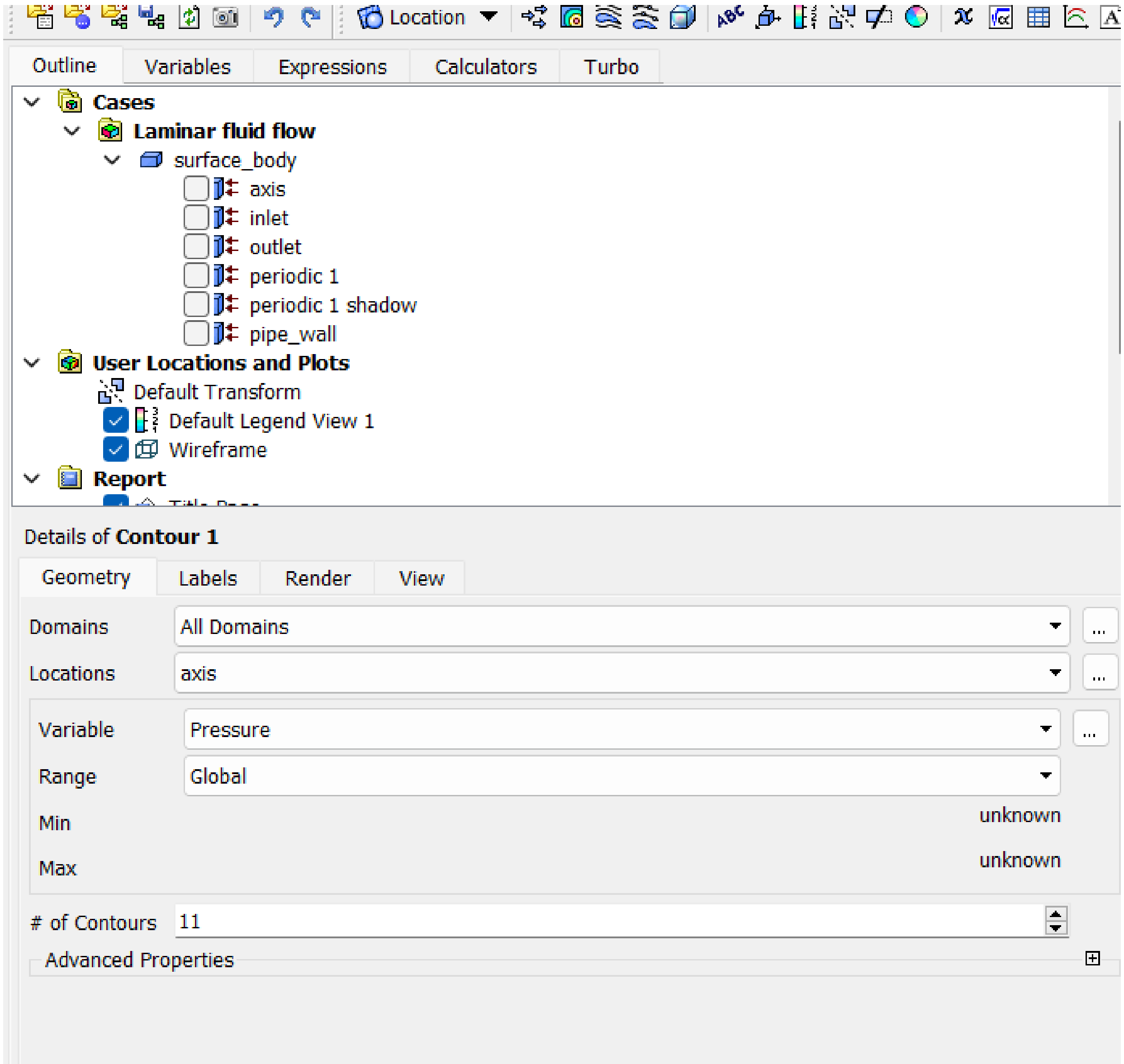


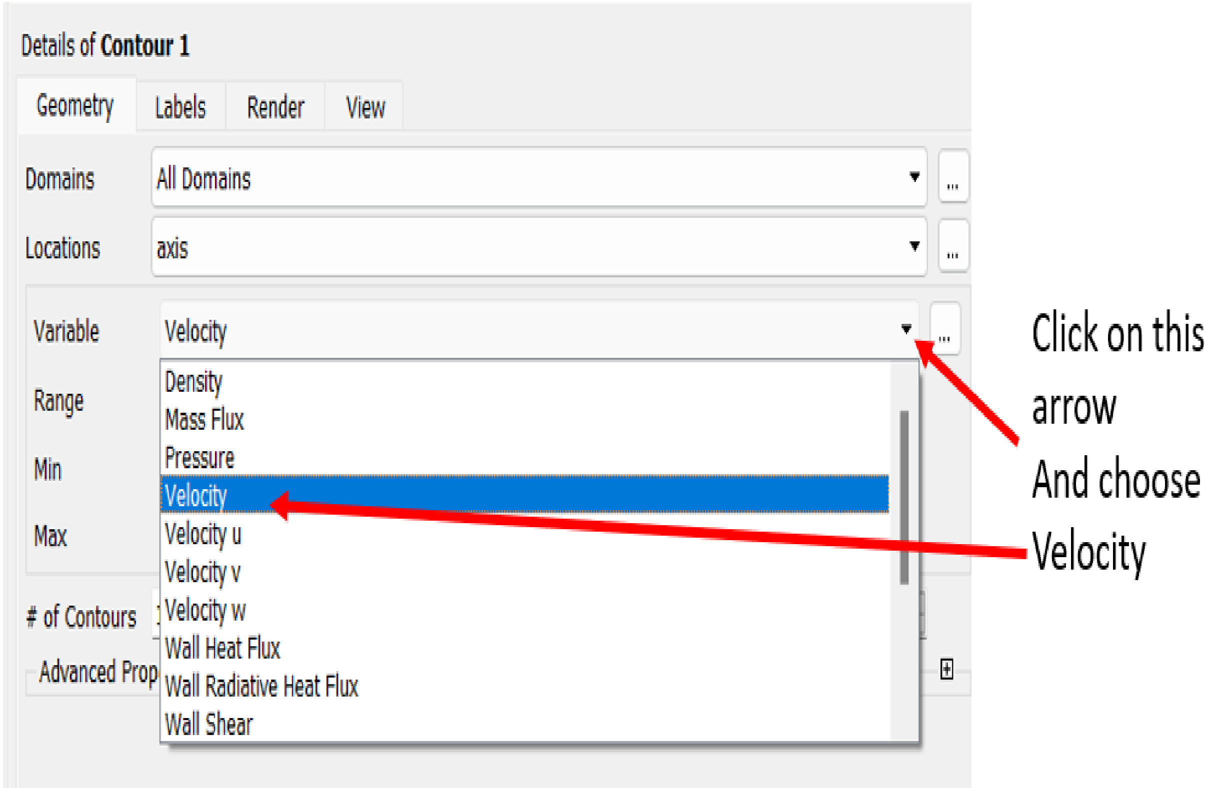
Click on
"Contour"
symbol on topo of PST
screen.
This box appears.

Click on "OK"

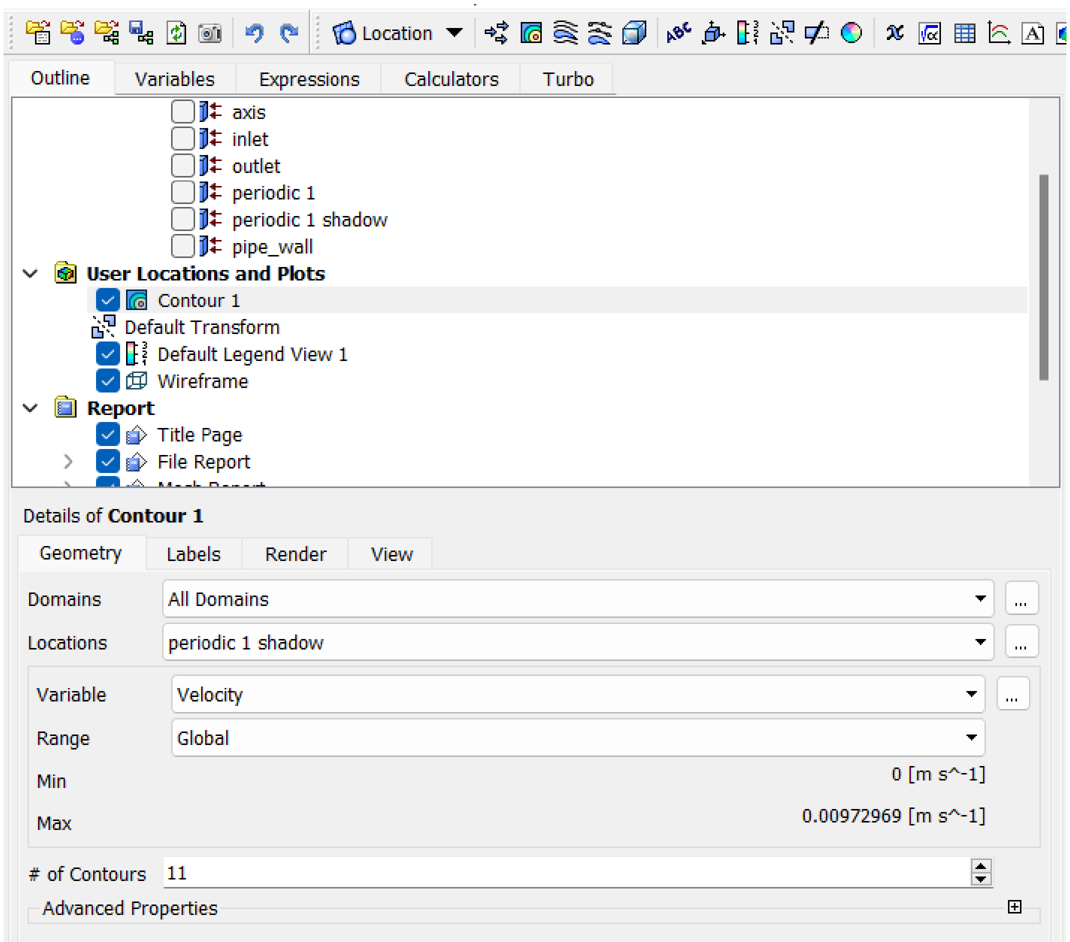


After clicking on "OK" above, the following "Contour" box appears on the lower left part of the PST window.

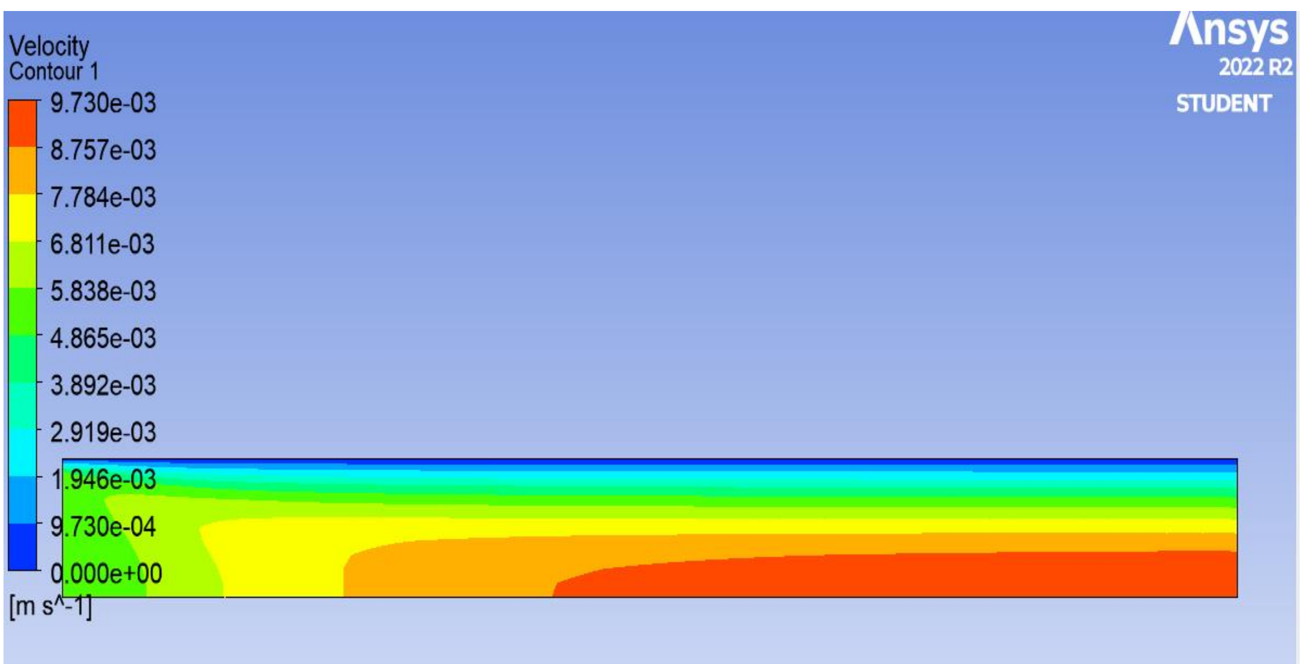




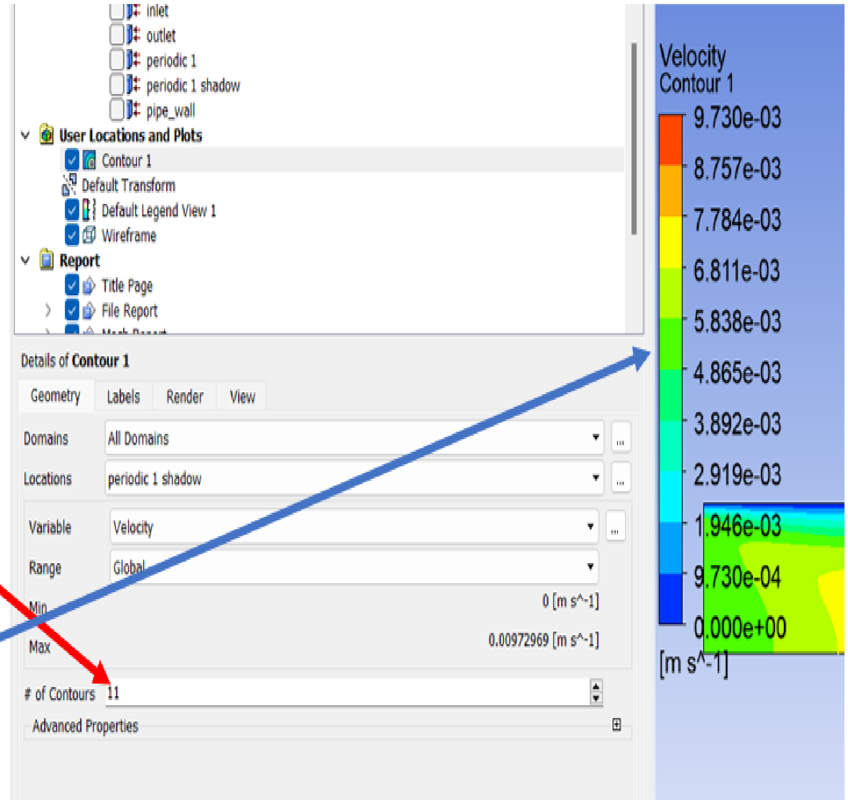
The choices for the "Contour 1" should be as shown below. Be careful with the choices. If the choices are not exactly as shown below, the contour will be blank.



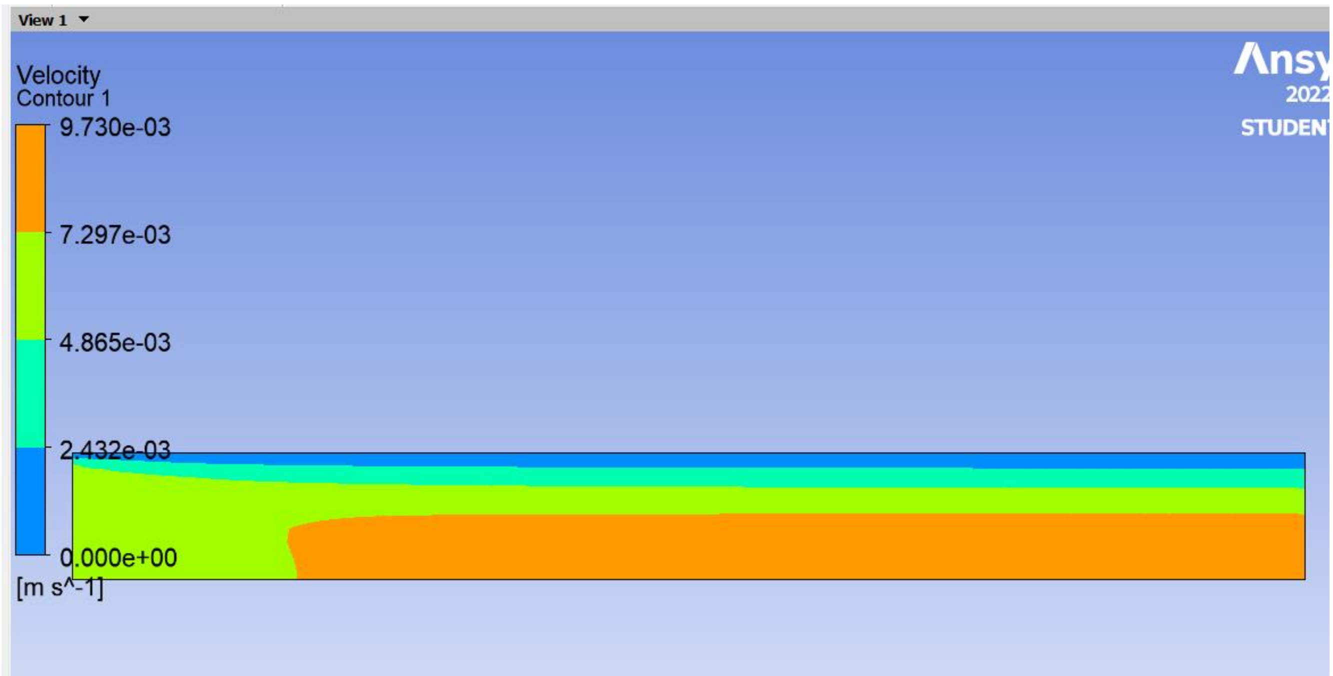
Click on “Apply” at the bottom left side of screen. The velocity profile appears as shown below.



This denotes the number of color bands in the contour. This number is indicated as “11”, but there are only 10 colors shown in the band.

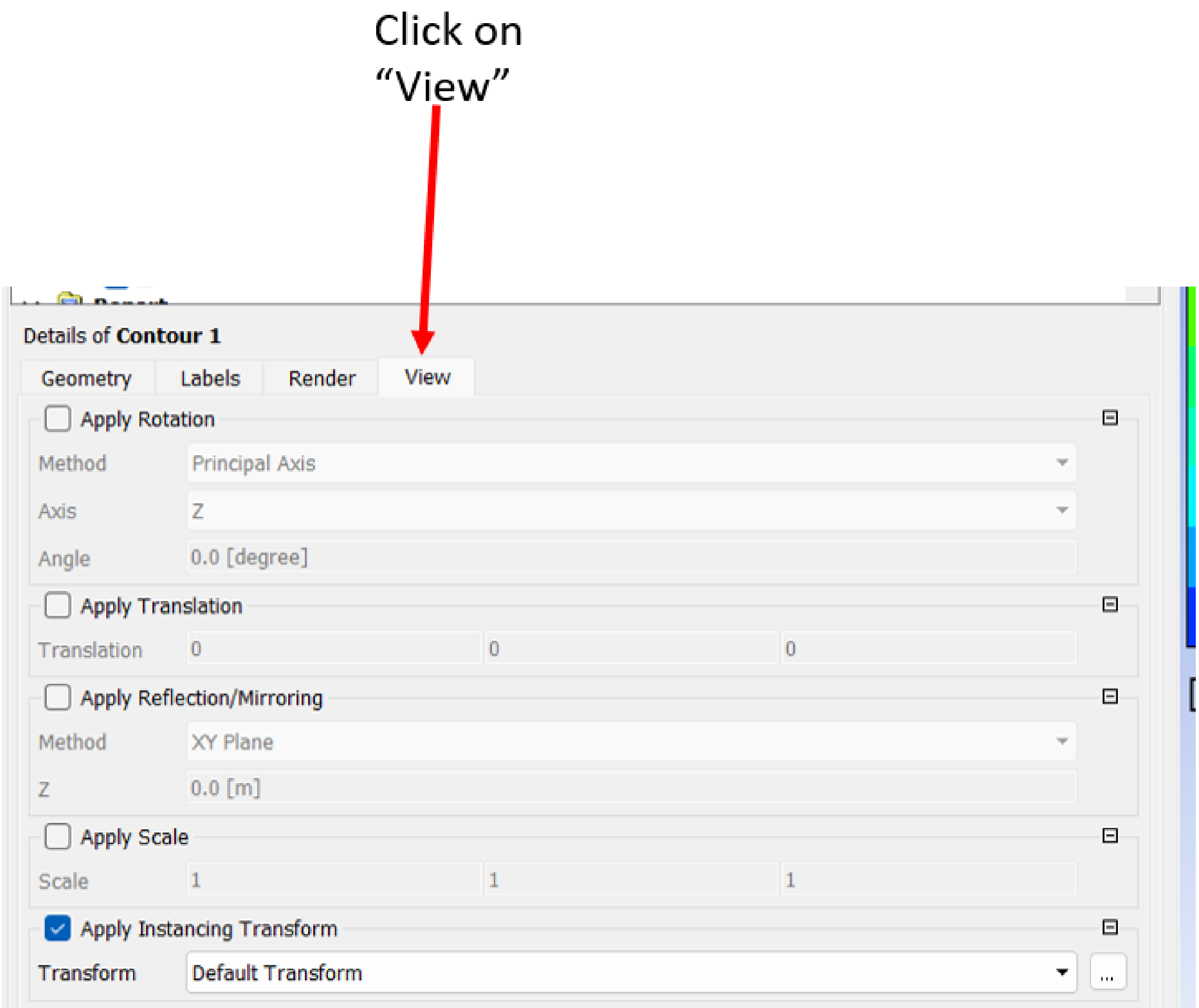


The number of colors can be changed by changing the 11 to 5, and clicking on "Apply" at the screen bottom. If the number of bands is changed to for example, to 5, the contour plot appears as shown below. As before, the number of colors is defined as "5", but only "4" colors are shown.

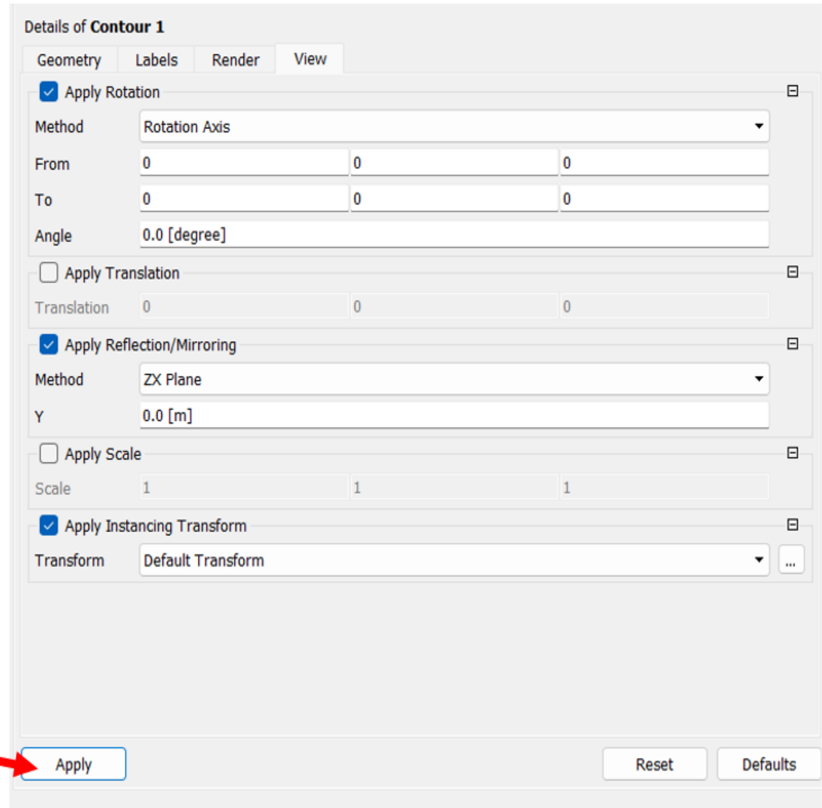


Recall that the model is axisymmetric which means the whole physical geometry is obtained by rotating the geometry around its axis by 360°.

To get the flow for the complete cross section, do the following steps.

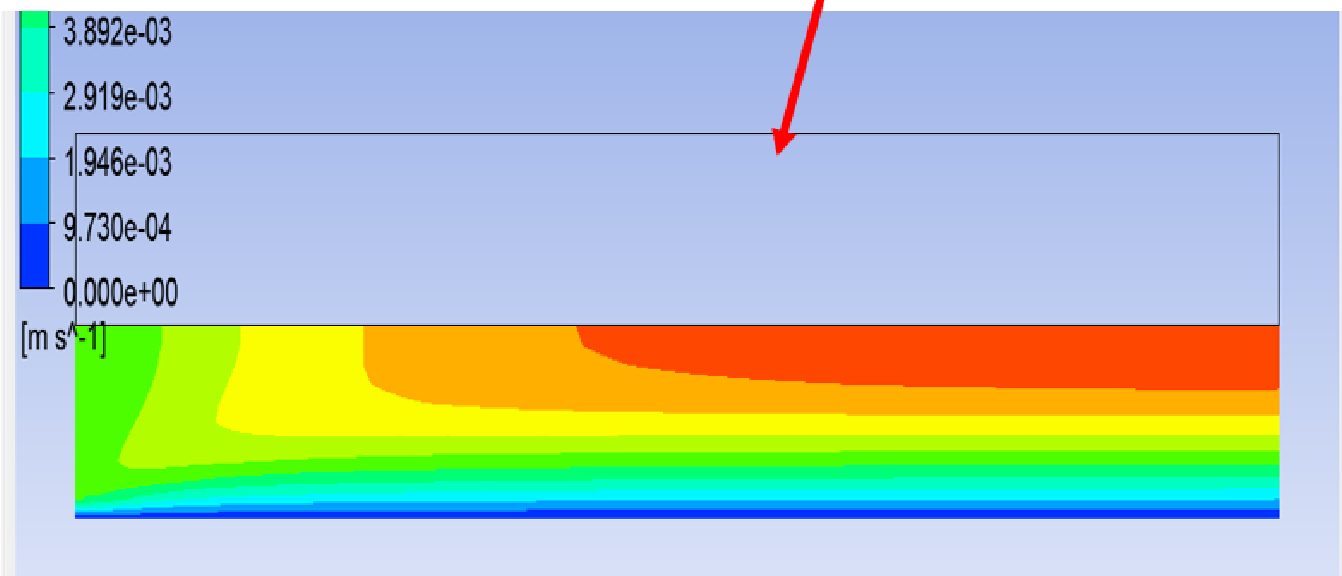


Under “View” tab, the EXACT choices shown below must be made.

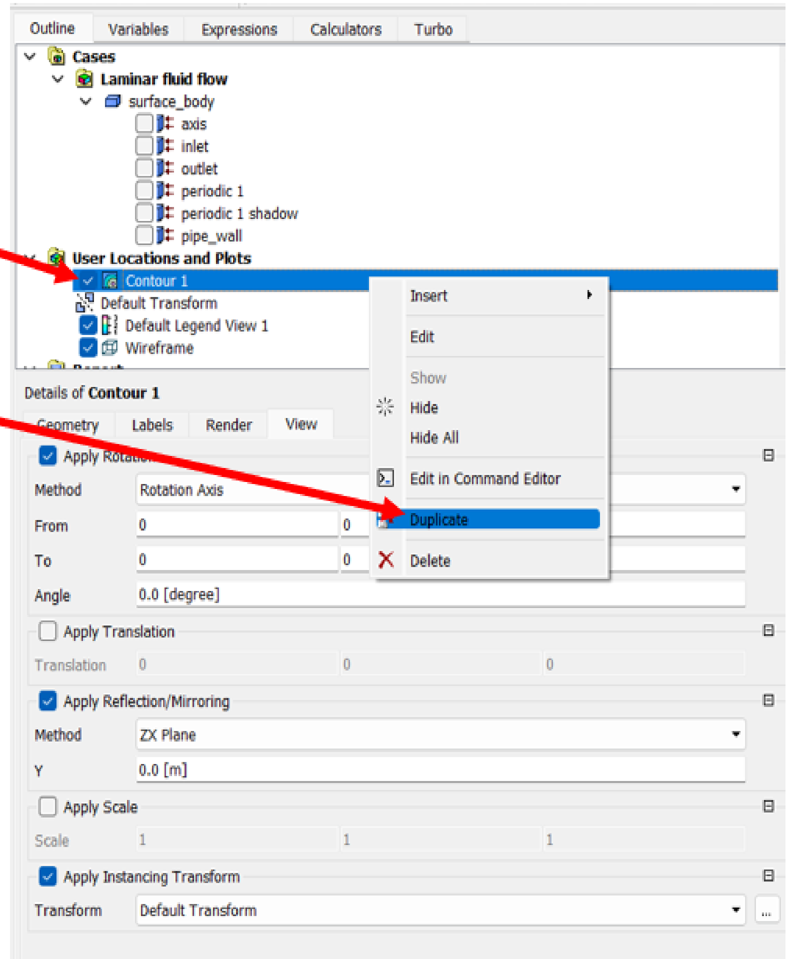


After the EXACT selections shown in the box are made, click on Apply

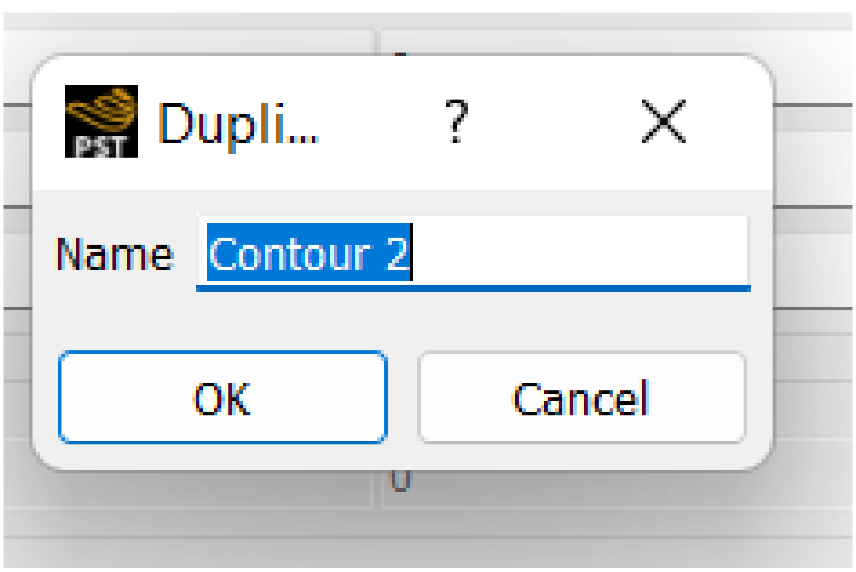
As a result of the previous operation, the blank mirror image shown is added to the screen.

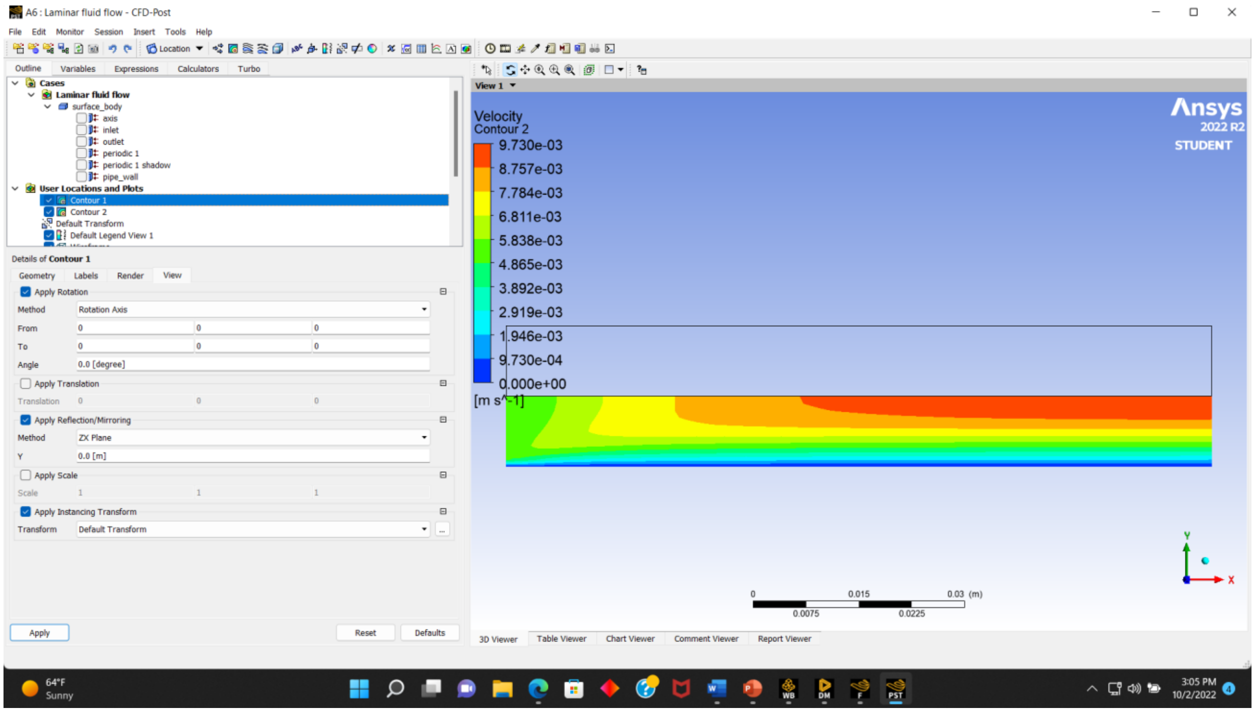


Right click on "Contour 1" And then click On "Duplicate"



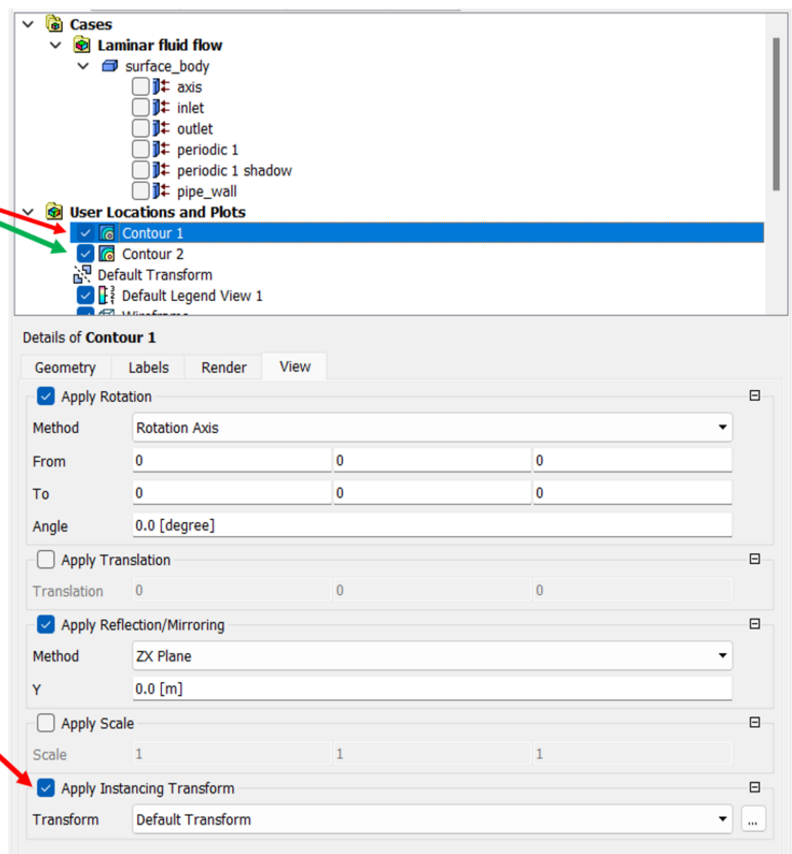
The following box appears. Click on "OK" in the following box.



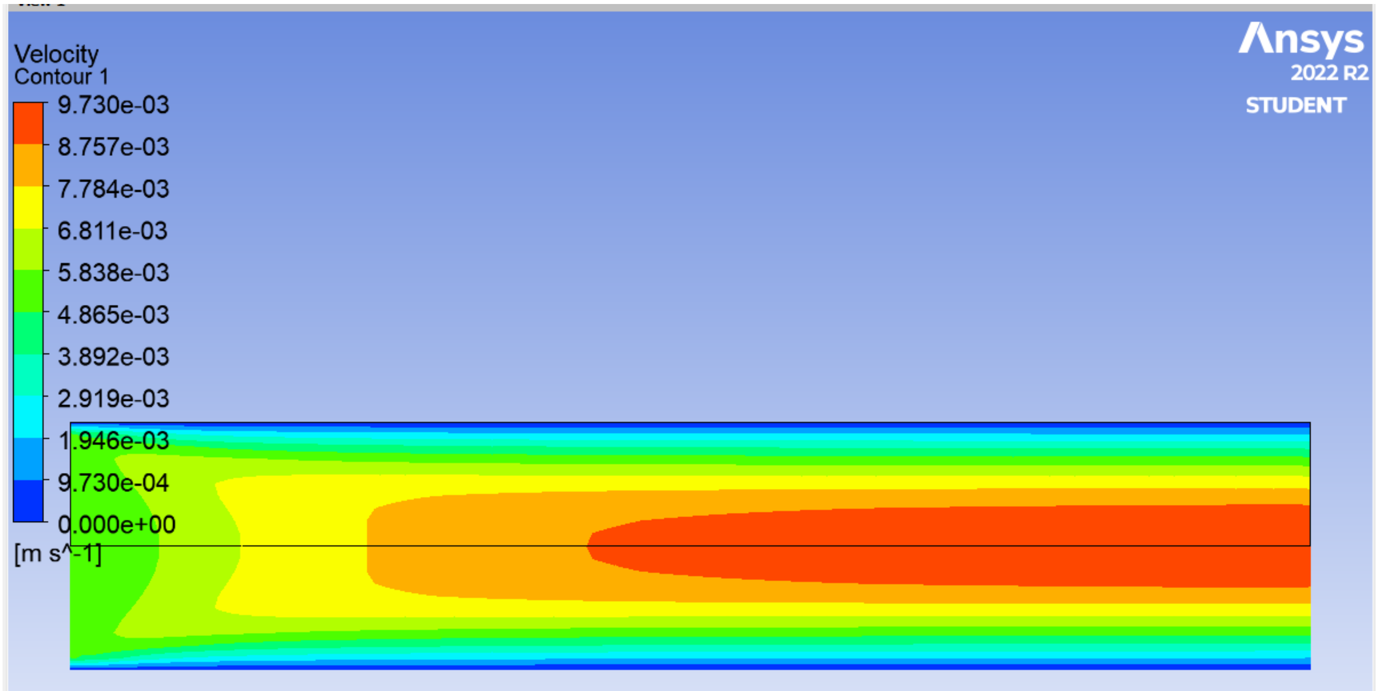


Notice that there are 2 contours now.

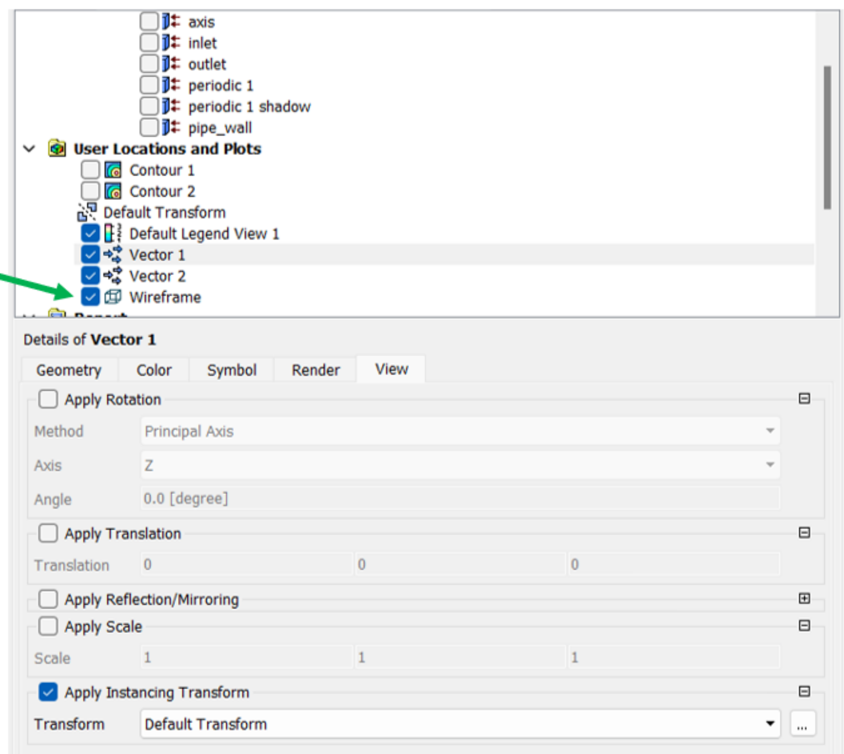
Uncheck the "Apply Reflection/Mirroring" and click on "Apply" at the bottom left of screen.



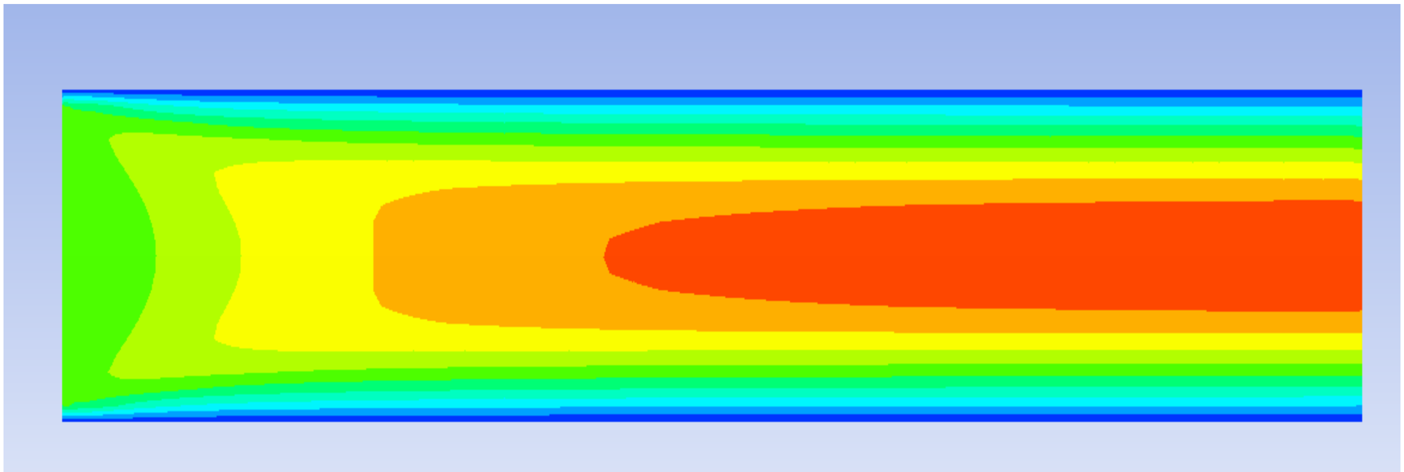
The complete velocity contour for the whole pipe cross section will appear as shown below.



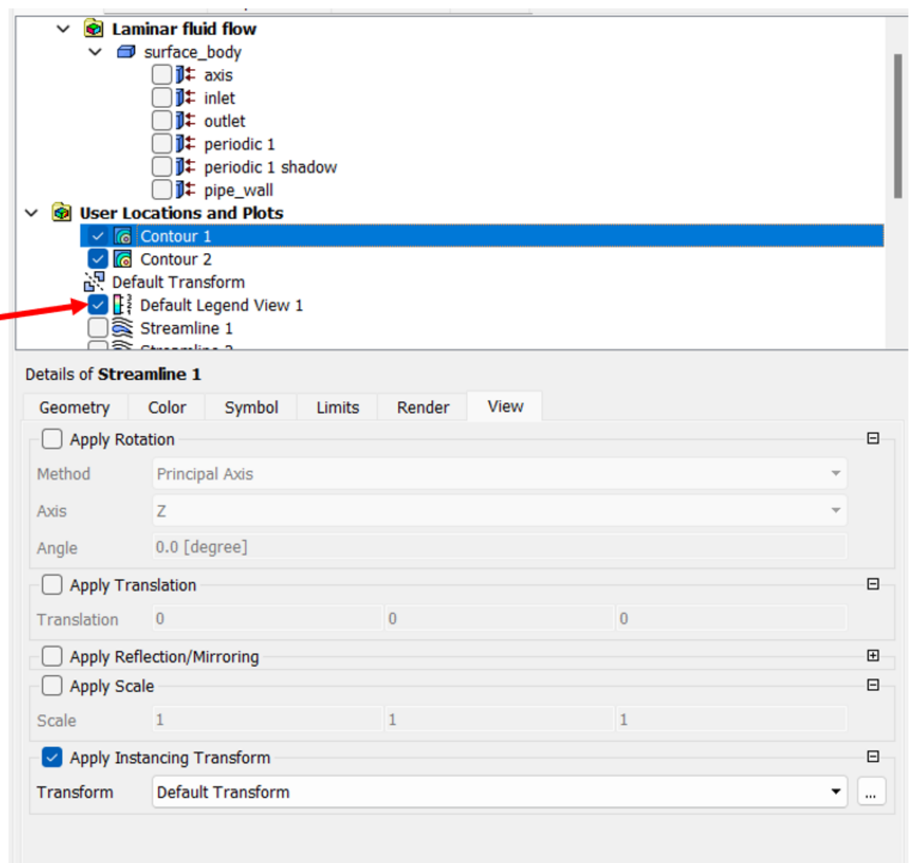
Uncheck the "Wireframe" to erase the box around half the graphic output.



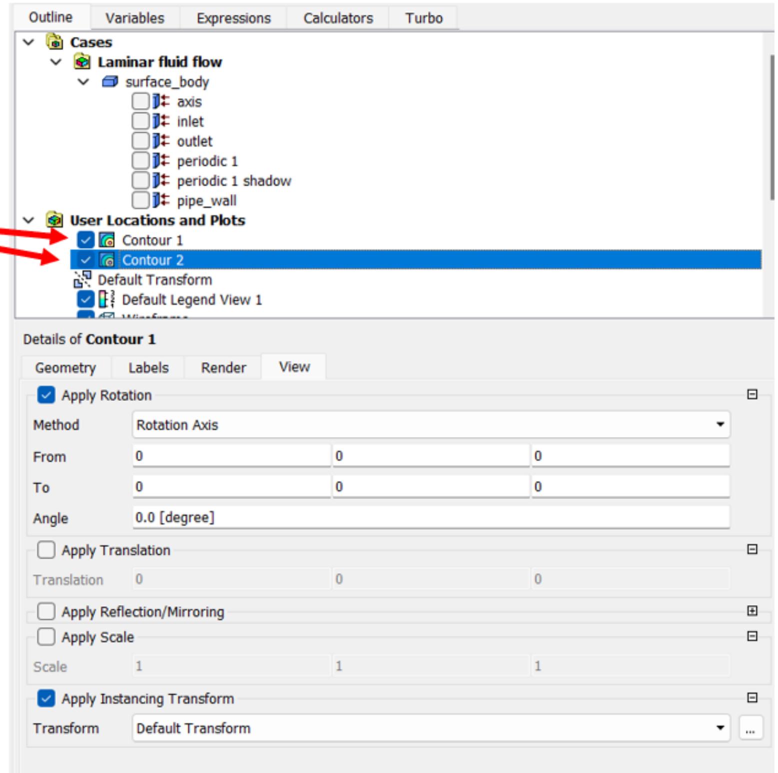
The following is the velocity contour without the box and without the ledger showing the values associated with each color.



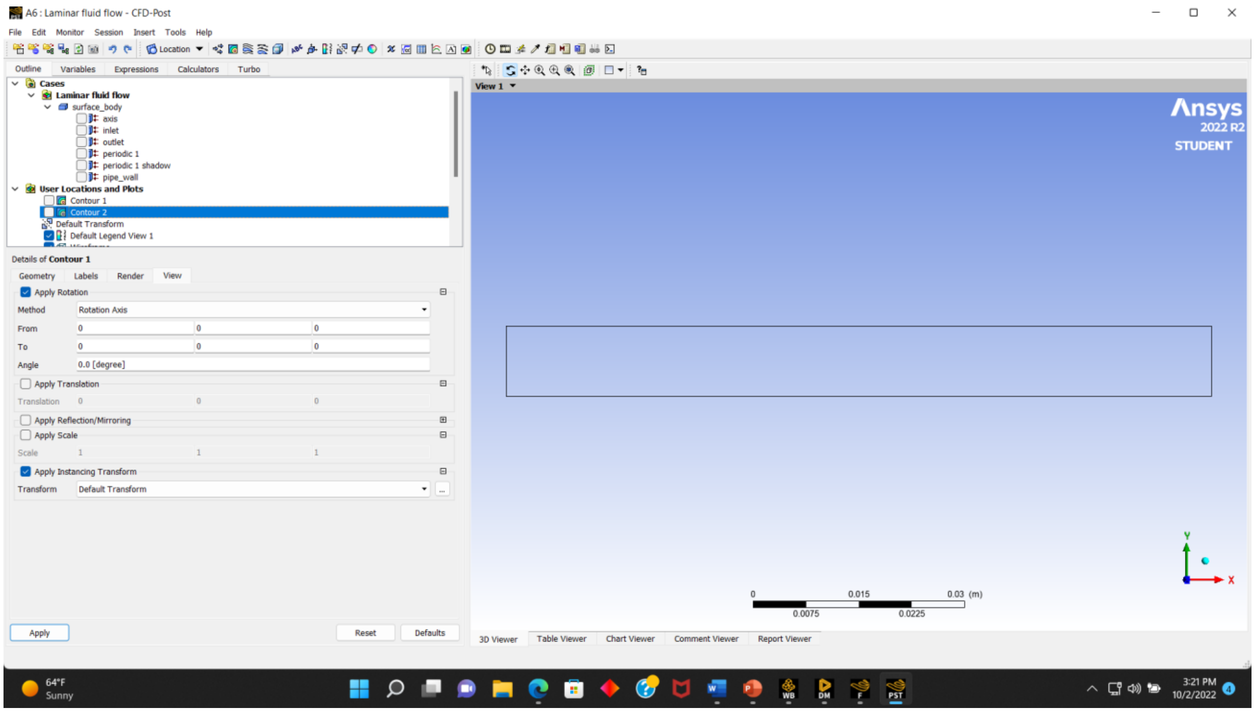
To make the ledger appear and disappear, click here to have the checkmark appear and disappear.



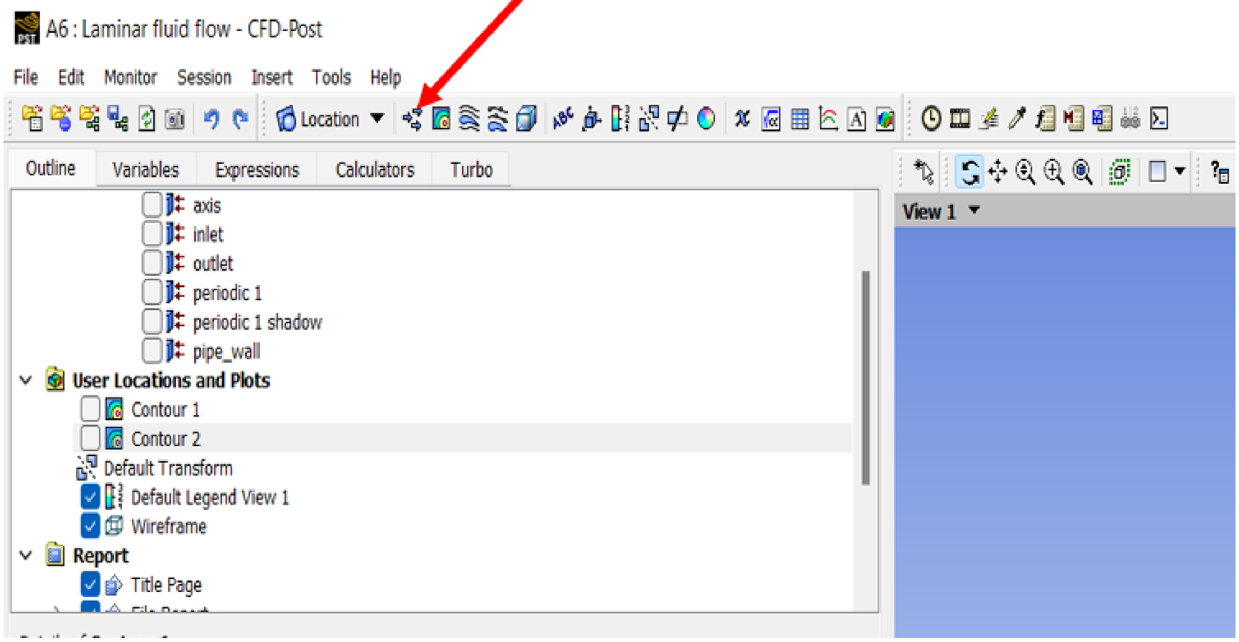
Uncheck these 2 boxes and the velocity profile becomes invisible.



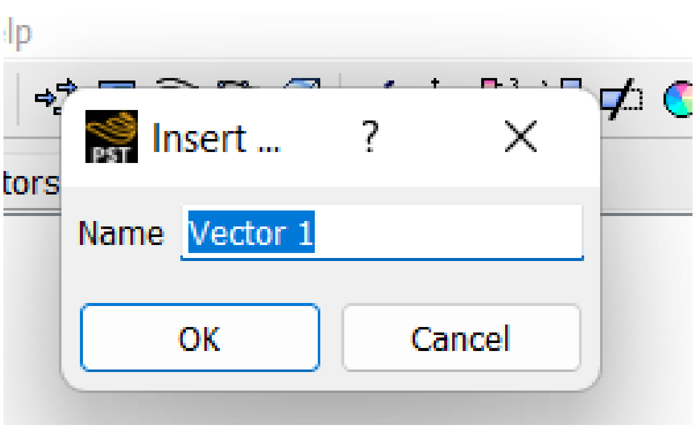
Below shows the screen after the velocity profiles have become invisible.



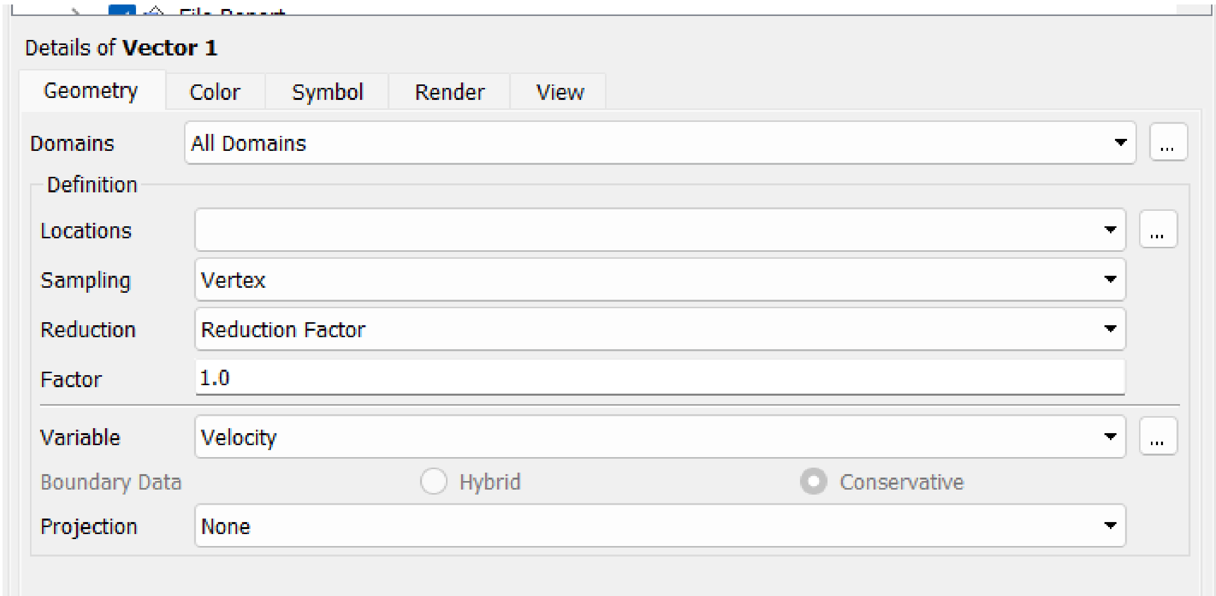
With the choices shown (exactly), click on “Velocity” tool.



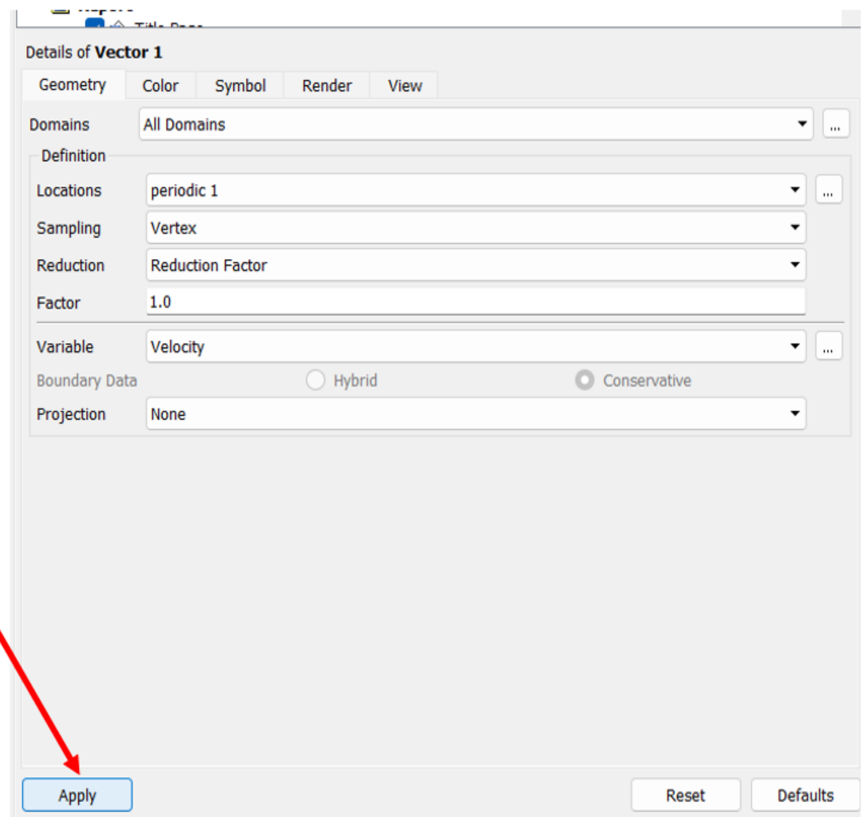
After clicking on “Velocity” above, the following pops up. Click on “OK”.



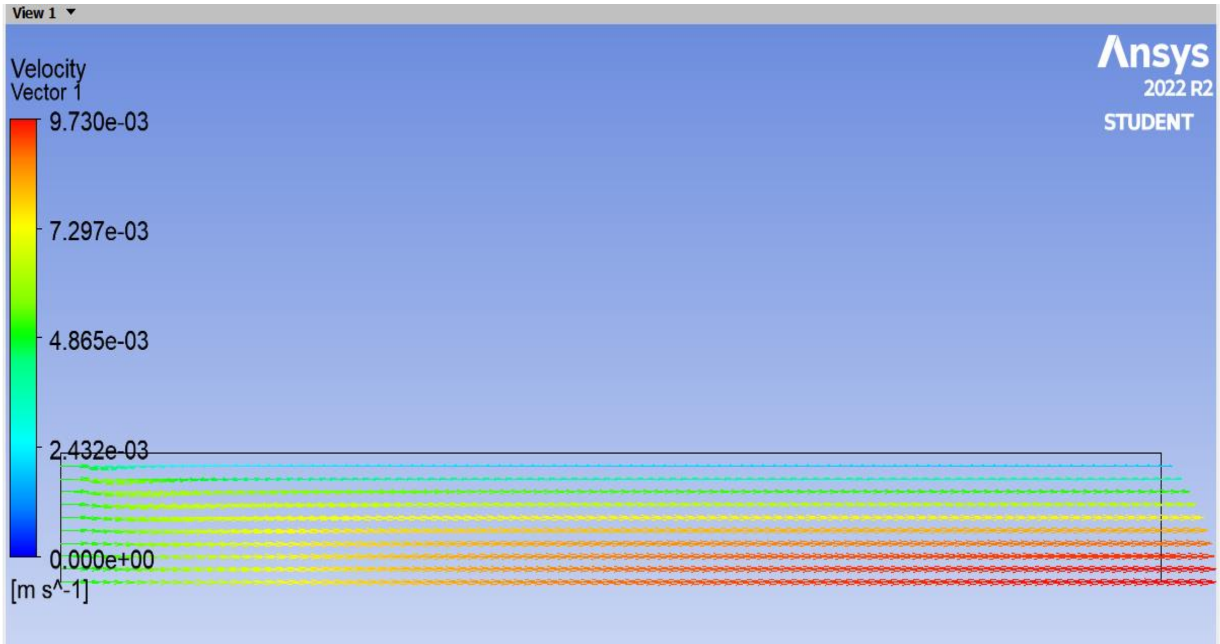
The following appears on the left side of screen.



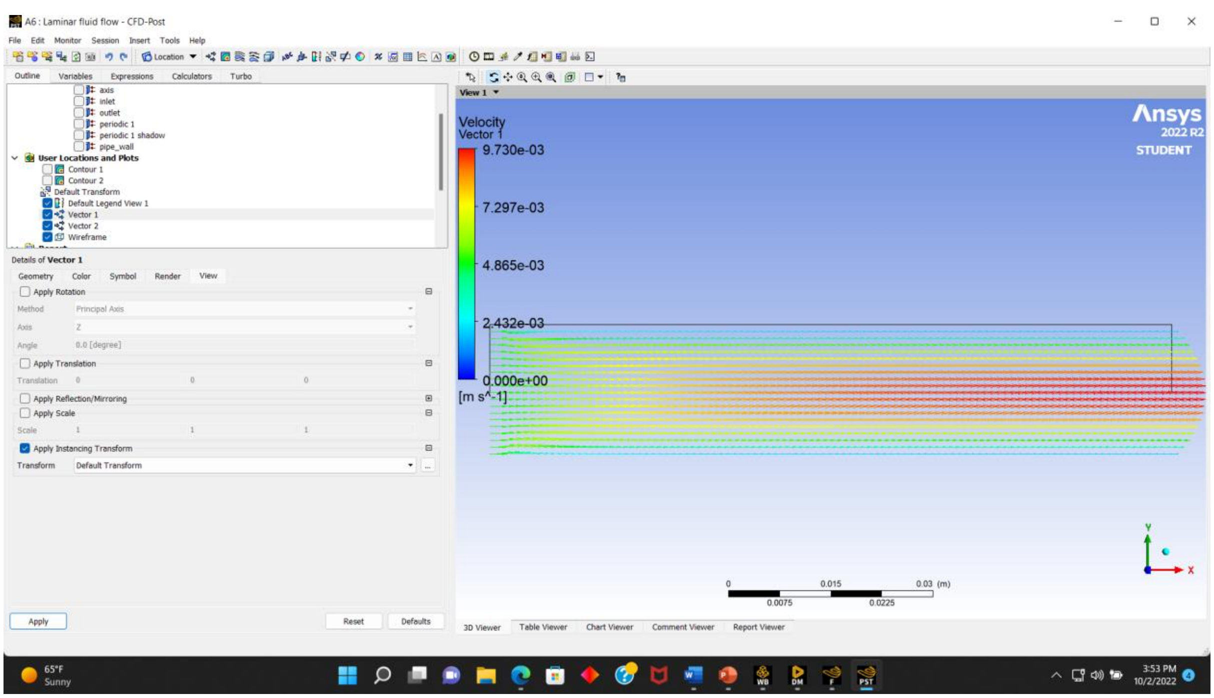
Make the choices shown and click on "Apply"



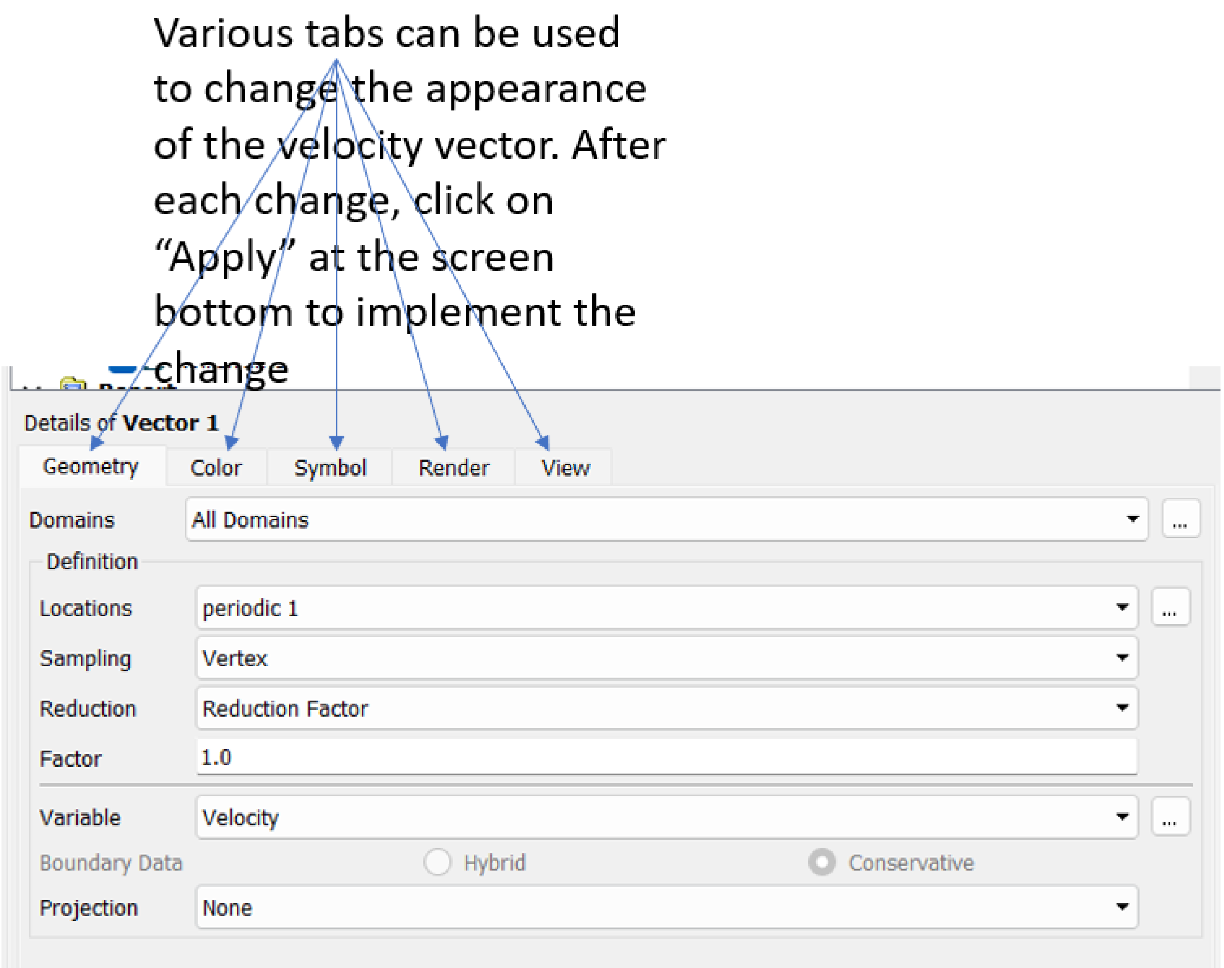
After clicking on “Apply”, a velocity vector is generated for half the pipe (because of axisymmetric modeling technique used). The following shows the model with the velocity vector.



By the same technique used for generating the velocity contour for the complete pipe cross section, the complete vector for the velocity can be generated. Make sure the name “Vector 1” and “Vector 2” which are the names of velocity vectors in the model are used in the procedure. The result showing the complete velocity profile for a cross section of the pipe is shown below.

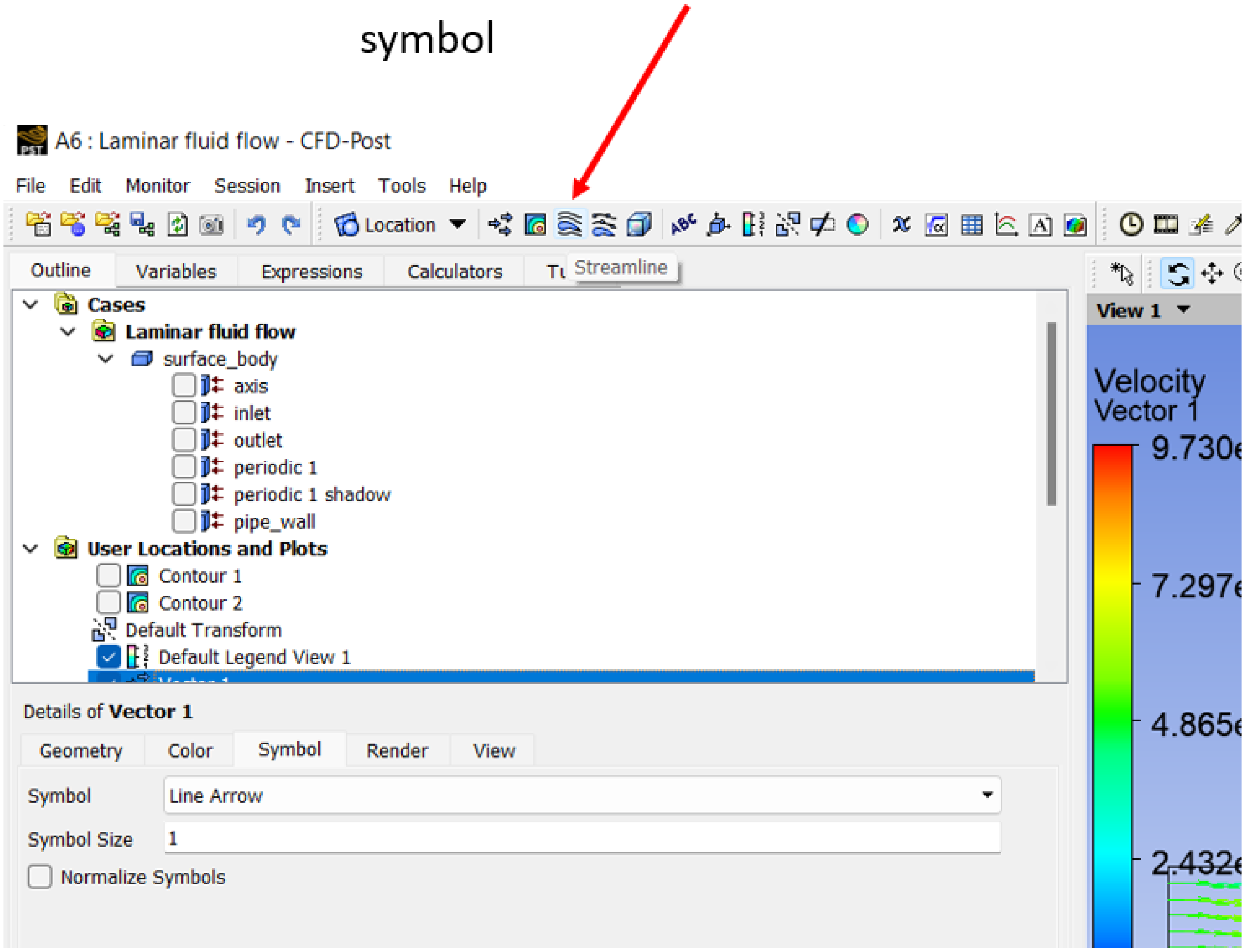


By unchecking the “Wireframe” on the left side of screen (it cannot be clearly seen above), the box around the upper portion of the box will disappear.

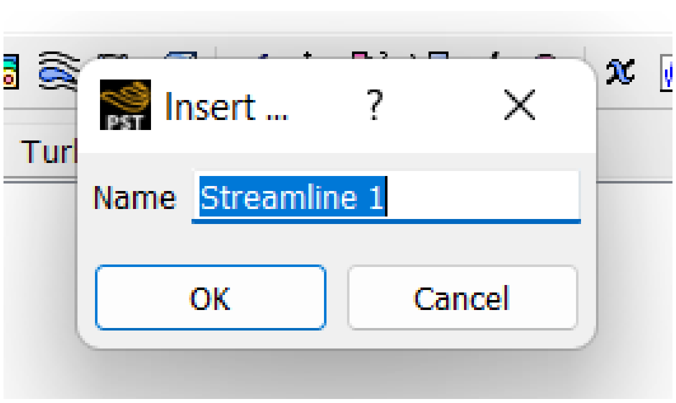


The velocity profiles can be made to appear and disappear by using the tree on the left side of the PST screen.

This is the “Streamline” symbol

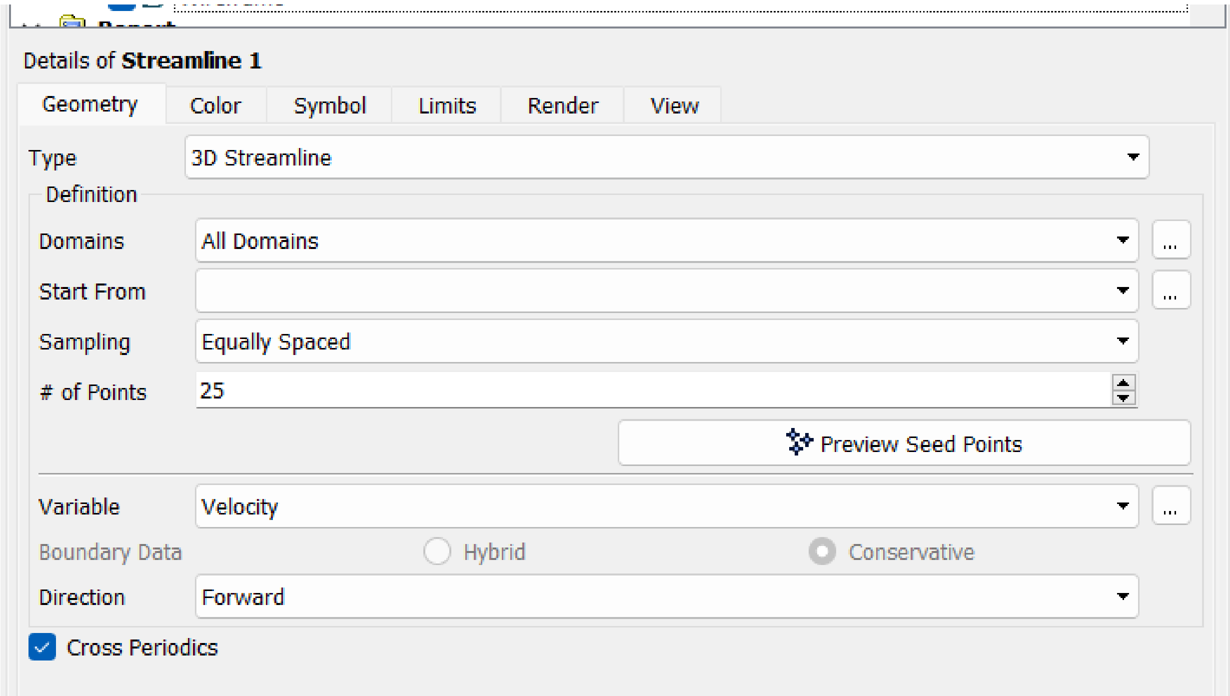


Click on the “streamline” symbol. The following appears after clicking on “Streamline”.



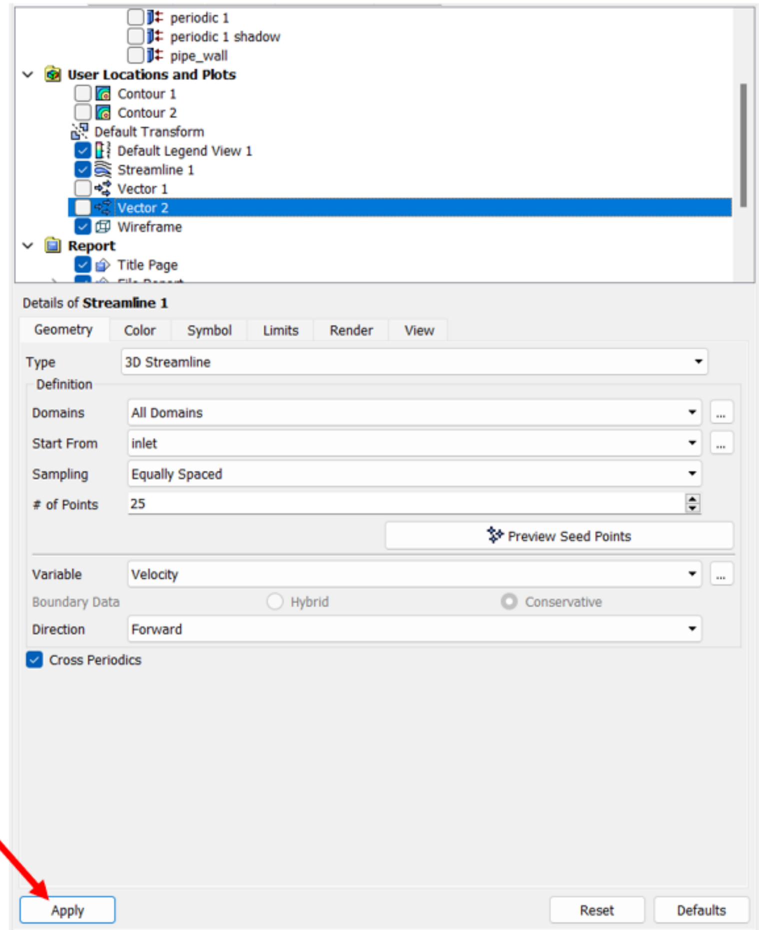
Click on “OK” above.

The following box appears where properties of the "Streamline" can be defined/modified.

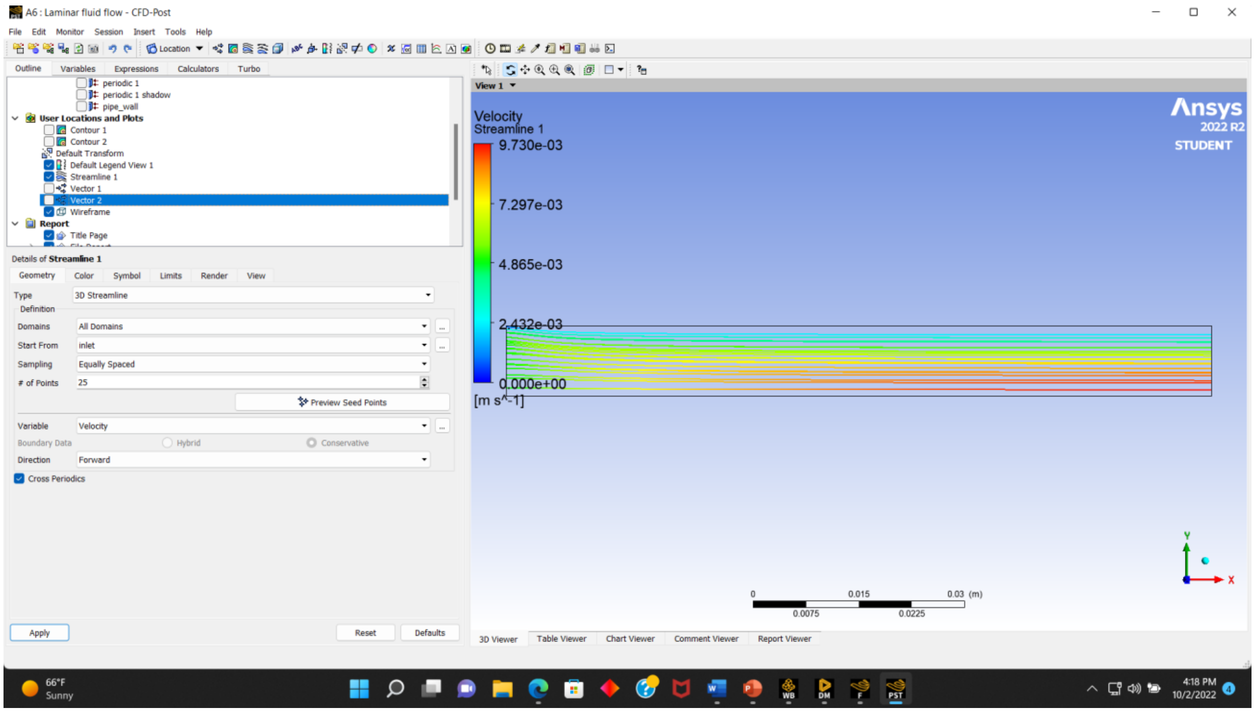


Make the velocity vectors to disappear by unchecking them on the tree on the left of PST screen.

Make the choices shown, and click on Apply

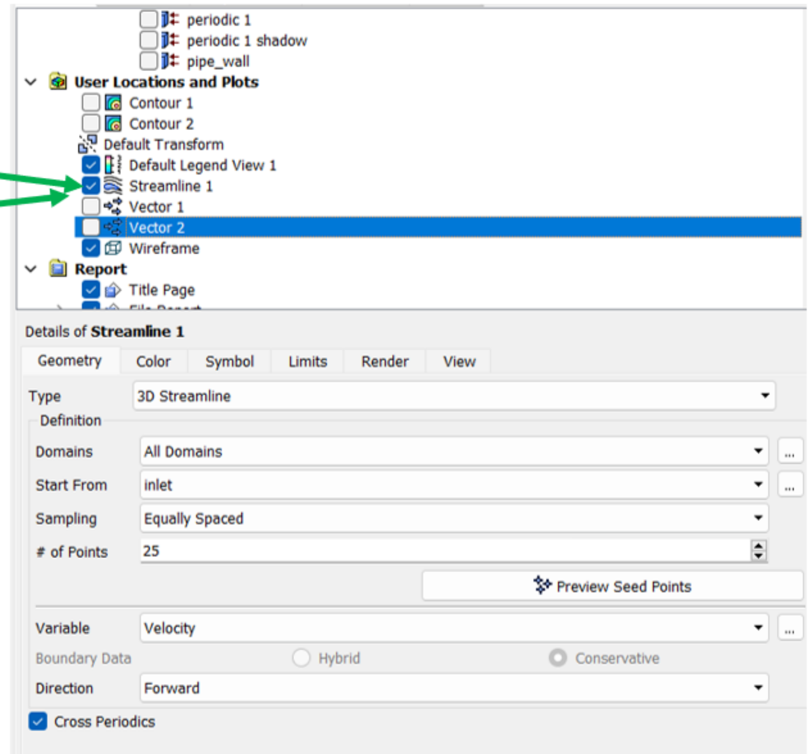


The streamline appears on the tree on the left of screen, and it will graphically display. The following shows the screen after “Apply” above is clicked on and the steam-line is generated.

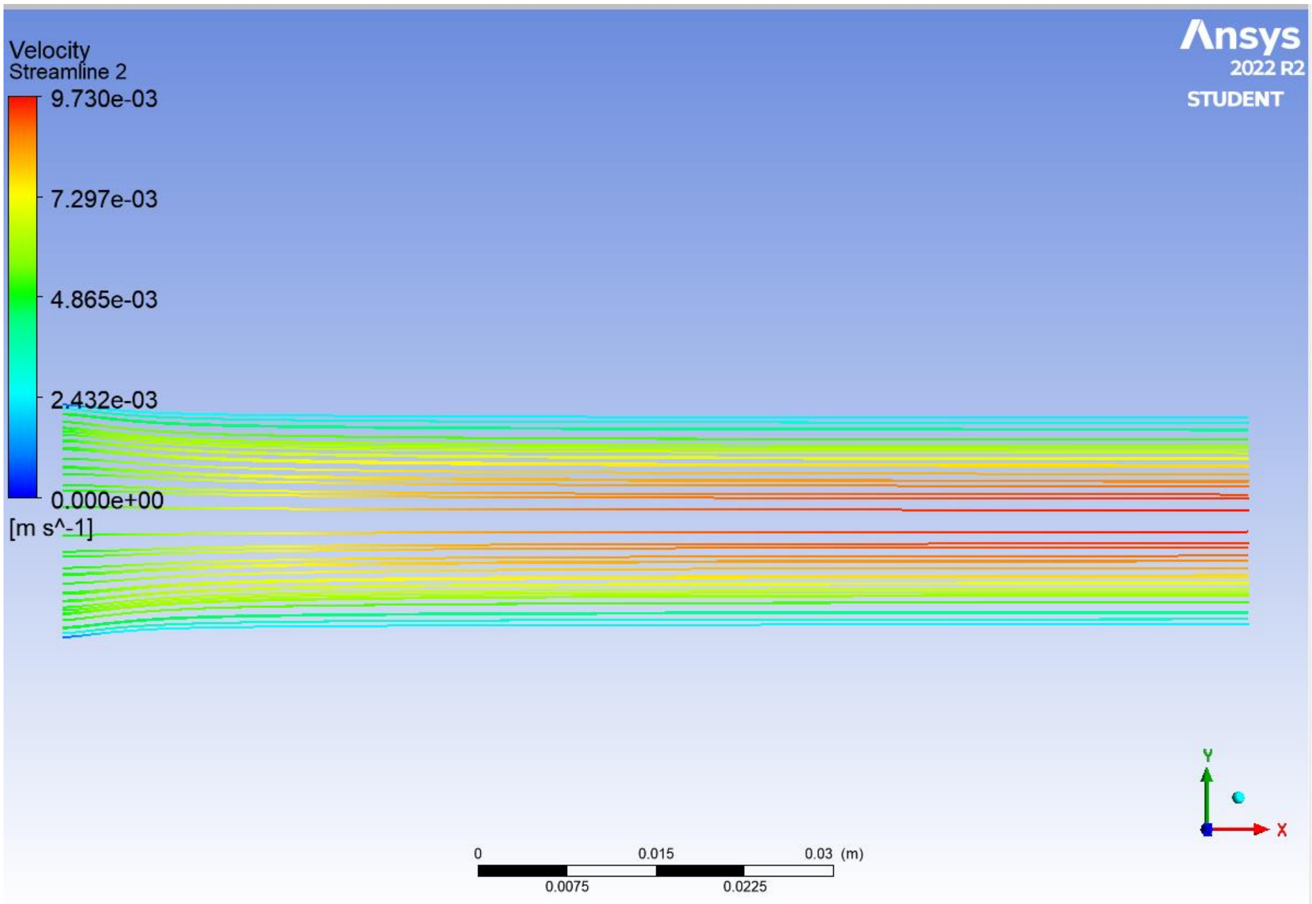


The following is the zoomed in tree showing the streamline has been added to the tree on the left of screen.

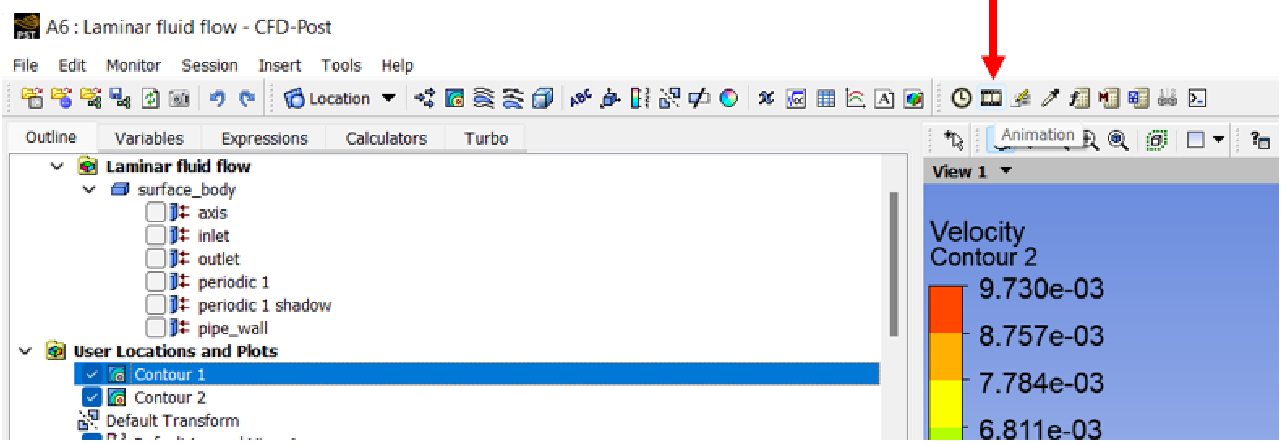
Streamline has been added "Check mark" is checked and consequently, the streamline is visible. If "check mark" is unchecked, the streamline will no be visible.



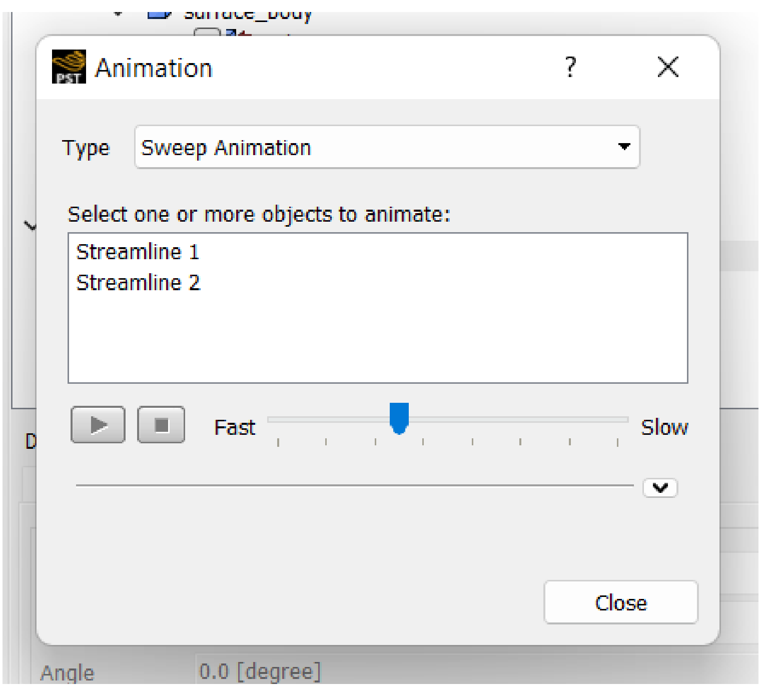
The streamline is for half the pipe cross section because the axisymmetric modeling technique is used. The complete pipe cross section can be generated by the technique used for getting the previous cross sections. The streamline for the complete cross section of the pipe is shown below.

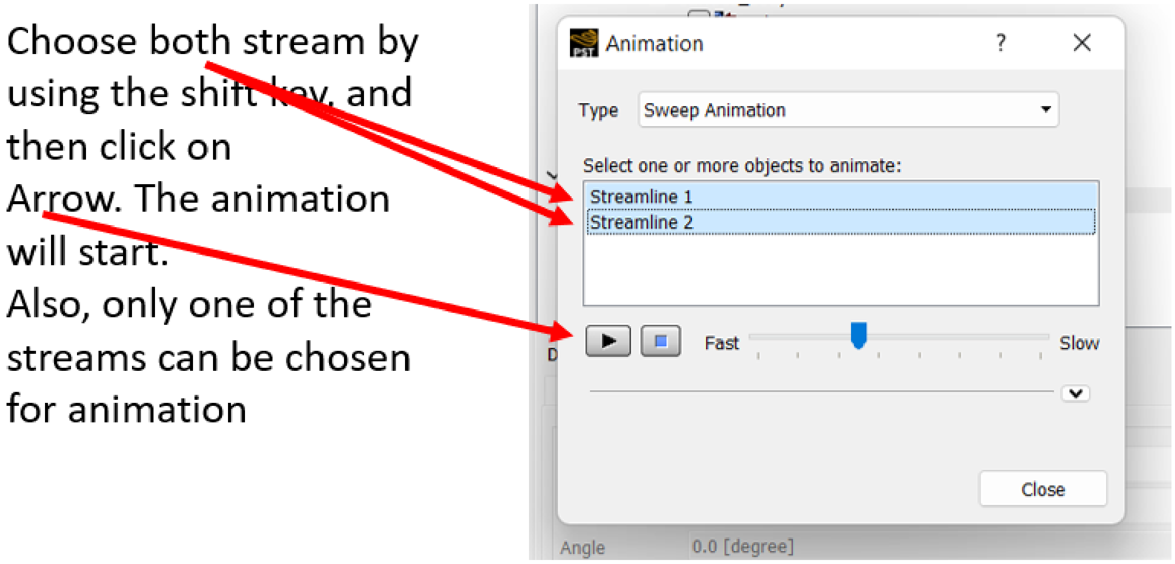


Click on the "Animation" Tool.



The following box appears.

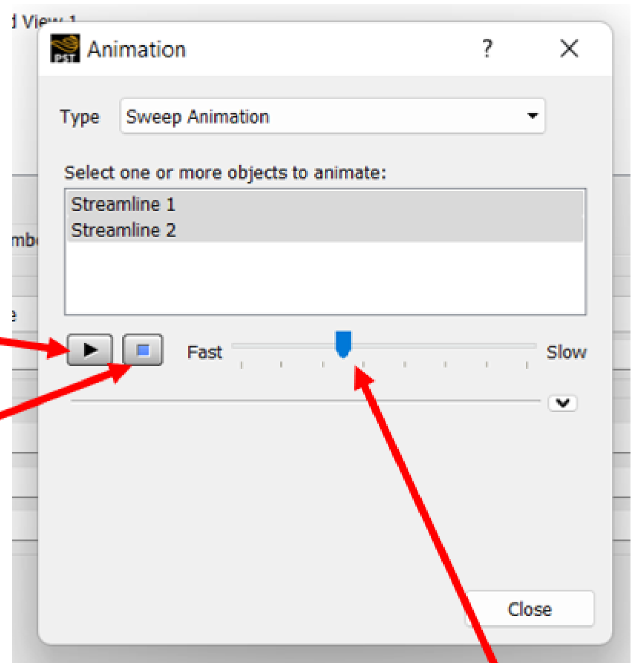




If animation does not work properly, make sure every other variable (for example velocity) is made invisible on the PST tree, and make sure “streamline 1” and “streamline 2” are visible by check marking them on tree, and then click the arrow above. If it still doesn’t work (the above animation won’t work), make the following selections.

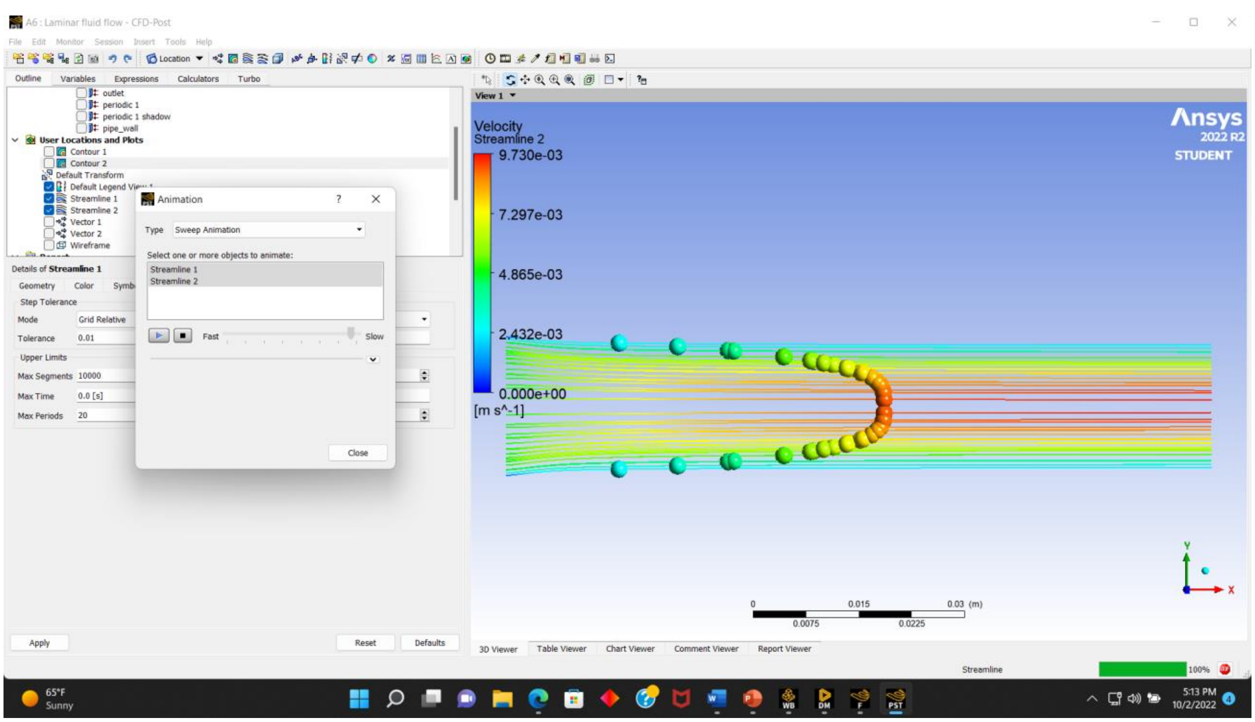
Click here to start the animation.

Click here to stop the animation



Animation speed can be adjusted by moving this.

The following is a screen shot of the animation.



At this point, save the project in the “WB” screen.

III: Summary and conclusion

Flow of water in a pipe was simulated using the CFD capabilities of ANSYS Work Bench. Step by step instructions have been provided in order to help a student with little or no experience in the use of CFD software. The results match the experimental results obtained in a fluids lab. While the technique is not technically complicated, it is beneficial to students that are new to fluid mechanics and CFD.

IV: References

1. Applied Fluid Mechanics; sixth edition by Robert L. Mott
2. ANSYS Work Bench manual
3. Ansys Fluent manual

V: Bibliography

Dr. Hagigat is a registered professional engineer with significant industrial experience is Aerospace, Commercial Nuclear Power and Petrochemical industry. He is currently a professor of Engineering Technology in a Midwestern United States University.