Environmental Transport Phenomena

Project – Part I: Tutorial

Laminar Pipe Flow

Mohammad Jamshidmofid
mohammad.jamshidmofid@epfl.ch

Fall 2025

Learning goals

Develop the numerical solution to a laminar pipe flow problem using Ansys Workbench, ANSYS SpaceClaim, ANSYS Meshing, ANSYS Fluent and ANSYS CFD-Post.

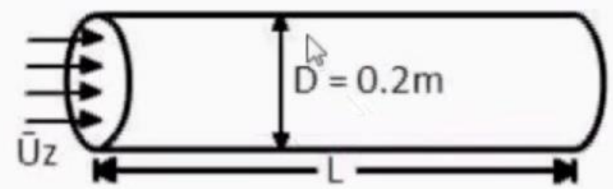
Assignment

Consider fluid flowing through a circular pipe of constant radius. The pipe diameter D = 0.2 m and length L = 8 m. Consider the inlet velocity to be constant over the cross-section and equal to 1 m/s. Take density $\rho = 1\ kg/m^3$ and coefficient of viscosity $\mu = 2 \times 10^{-3}\ kg/(m\ s)$. These parameters have been chosen to get a desired Reynolds number of 100.

Each student should work individually on the tutorial and must submit the requested files (.zip with workbench case file and folder and screen capture indicating the value of momentum balance shown in slide N° 71 of the tutorial).

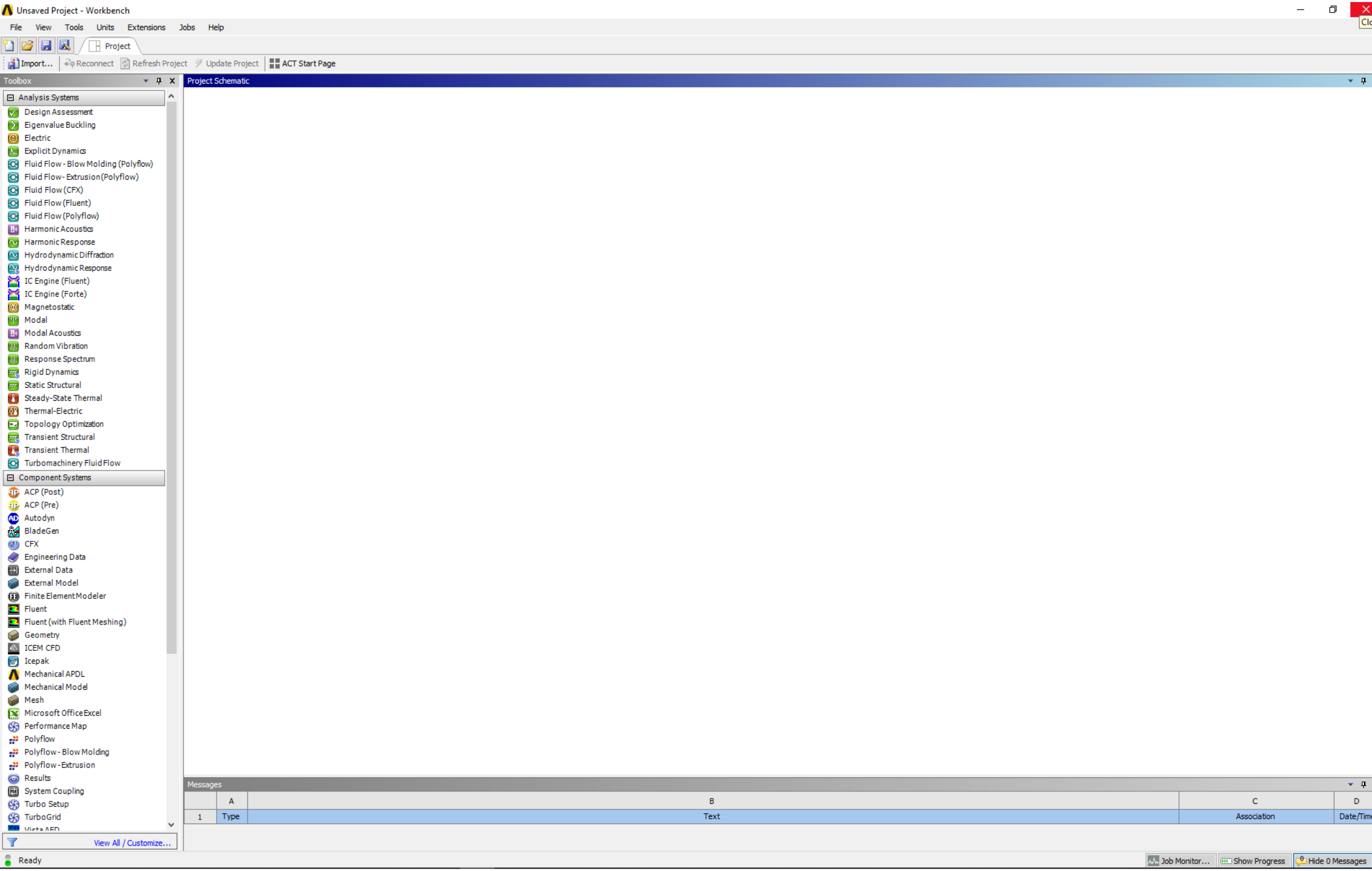
Summary of the assignment

Laminar Pipeflow

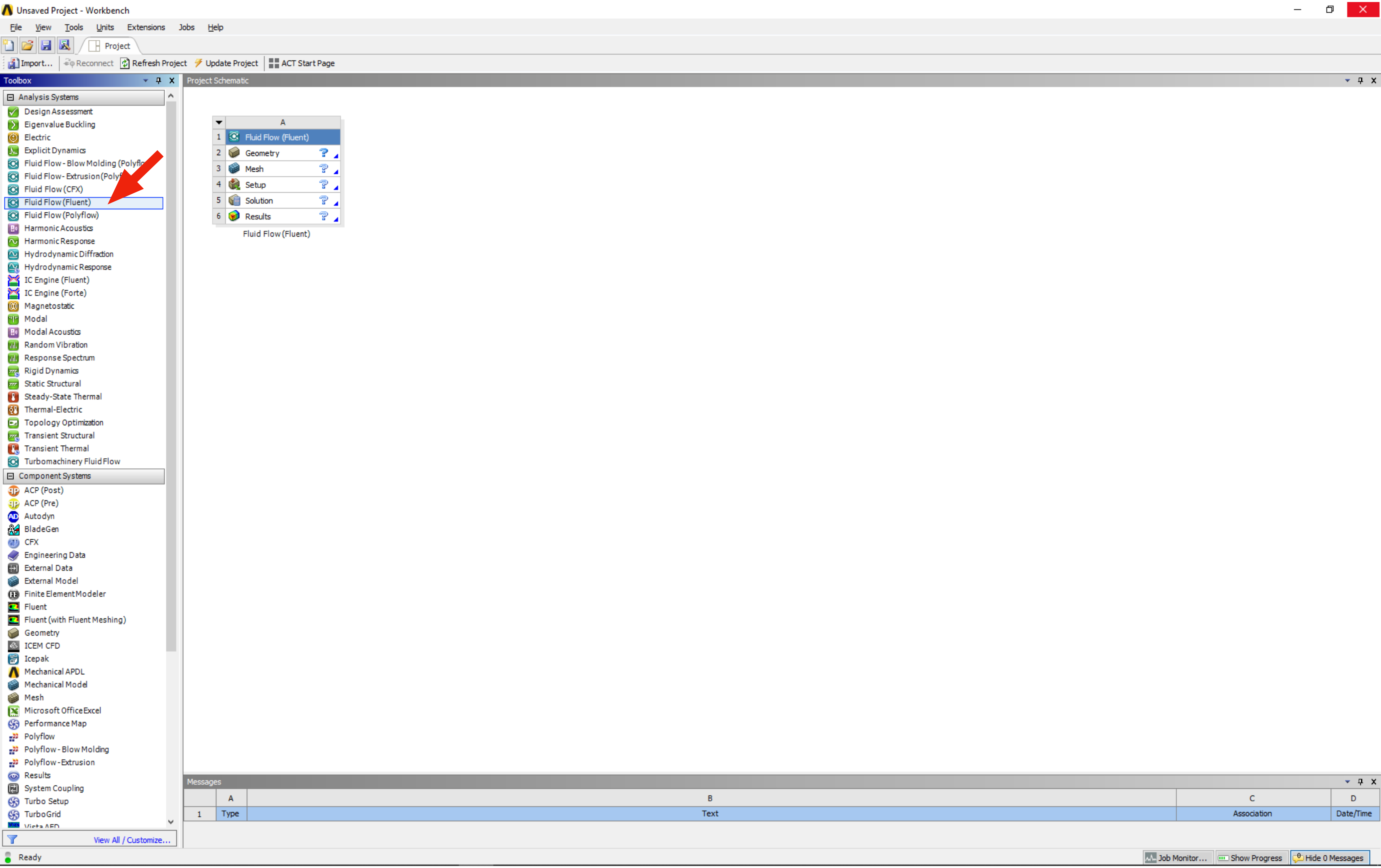


- Reynolds number of 100
  - $U = 1\ m/s$
  - $\rho = 1\ kg/m^3$
  - $\mu = 2 \times 10^{-3}\ kg/(m\ s)$.
- Geometry
  - $D = 0.2\ m$
  - $L = 8.0\ m$
- Produce
  - Velocity vectors
  - Velocity magnitude contours
  - Velocity profile at the outlet

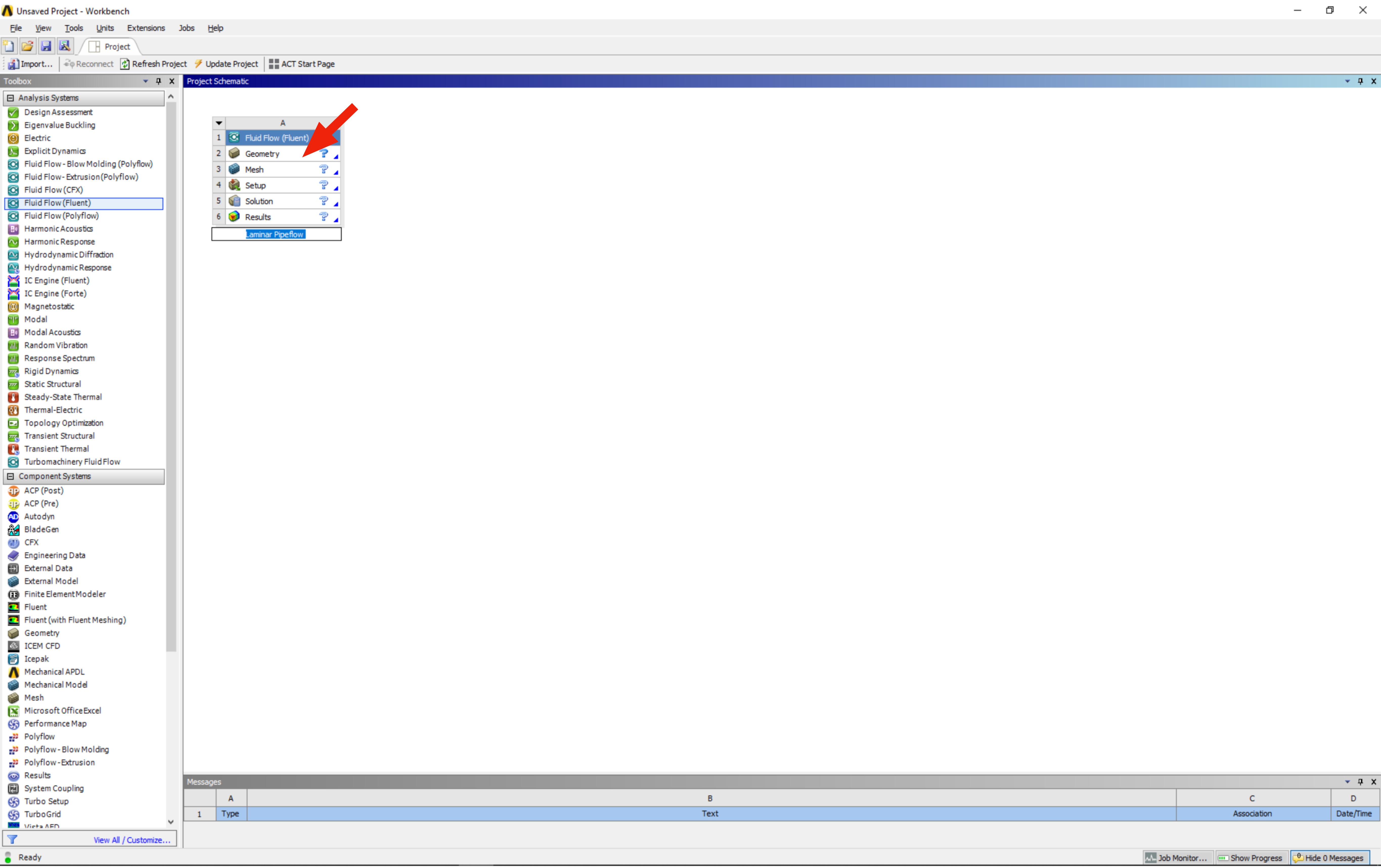
Run Workbench: (start - Workbench 2021 R1)



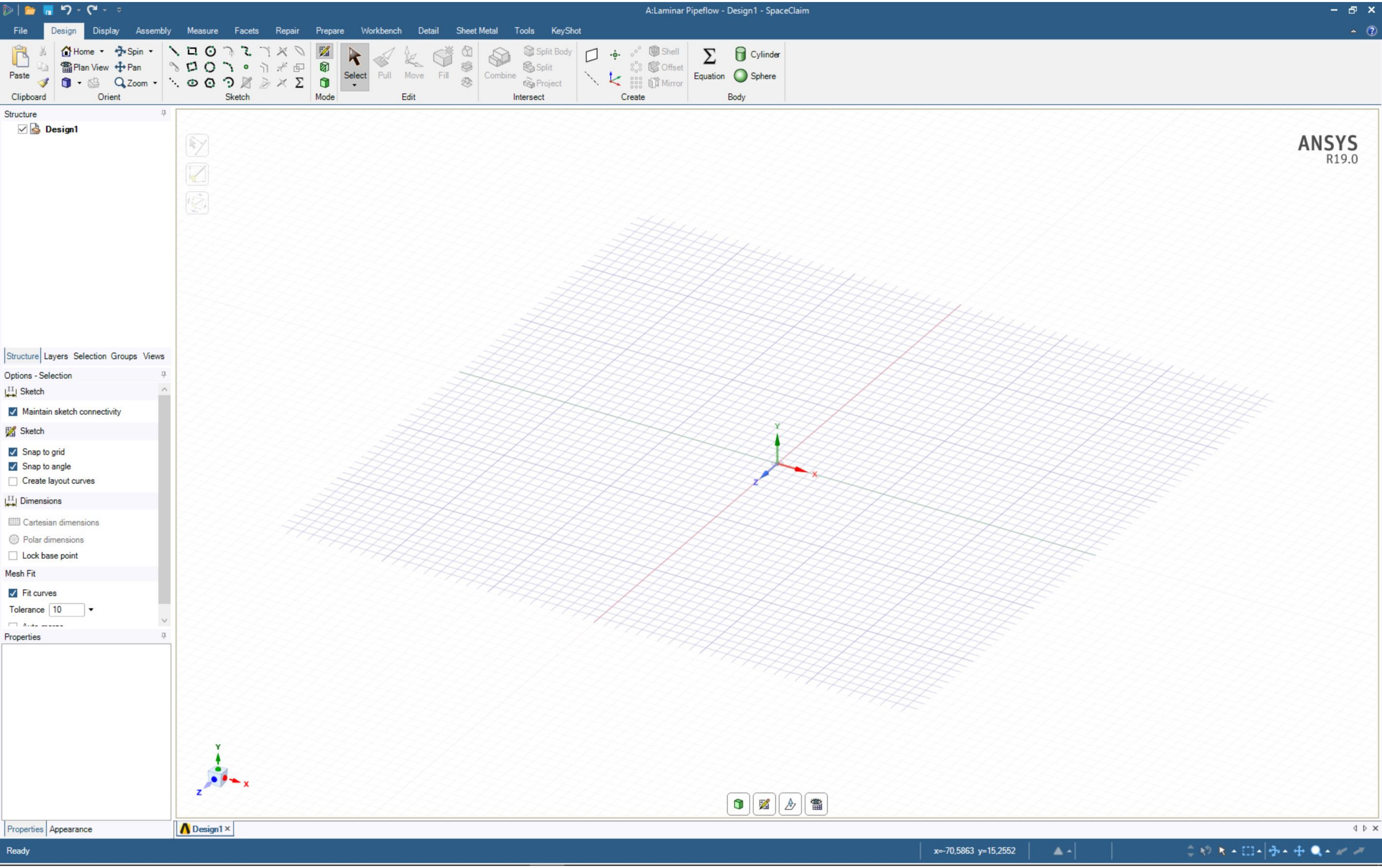
Drag the Fluid Flow icon to the Project Schematic window



Run Geometry (SpaceClaim Direct Modeler)

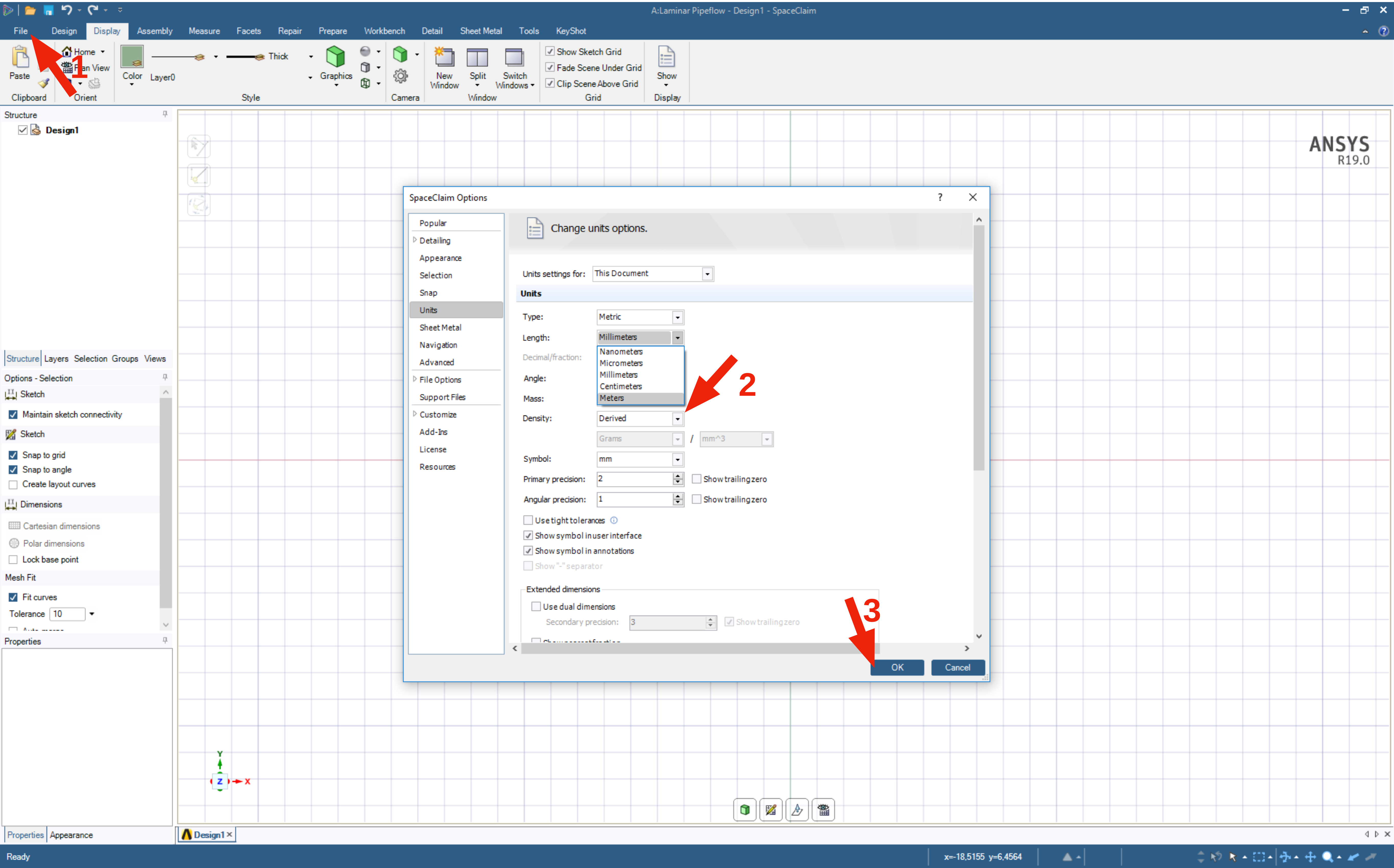


1- SpaceClaim Direct Modeler



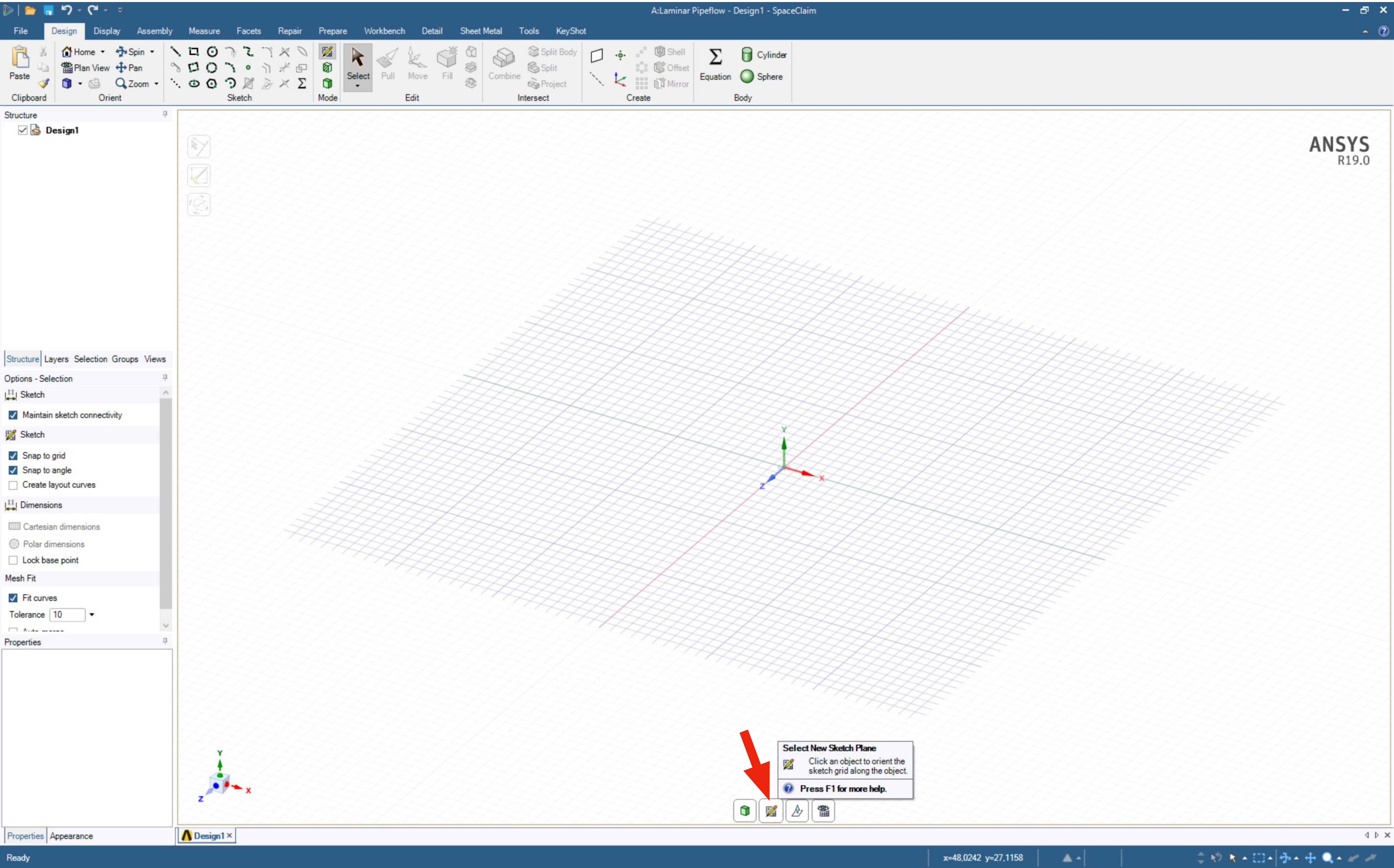
Change units in SpaceClaim

File/SpaceClaim Options/Units



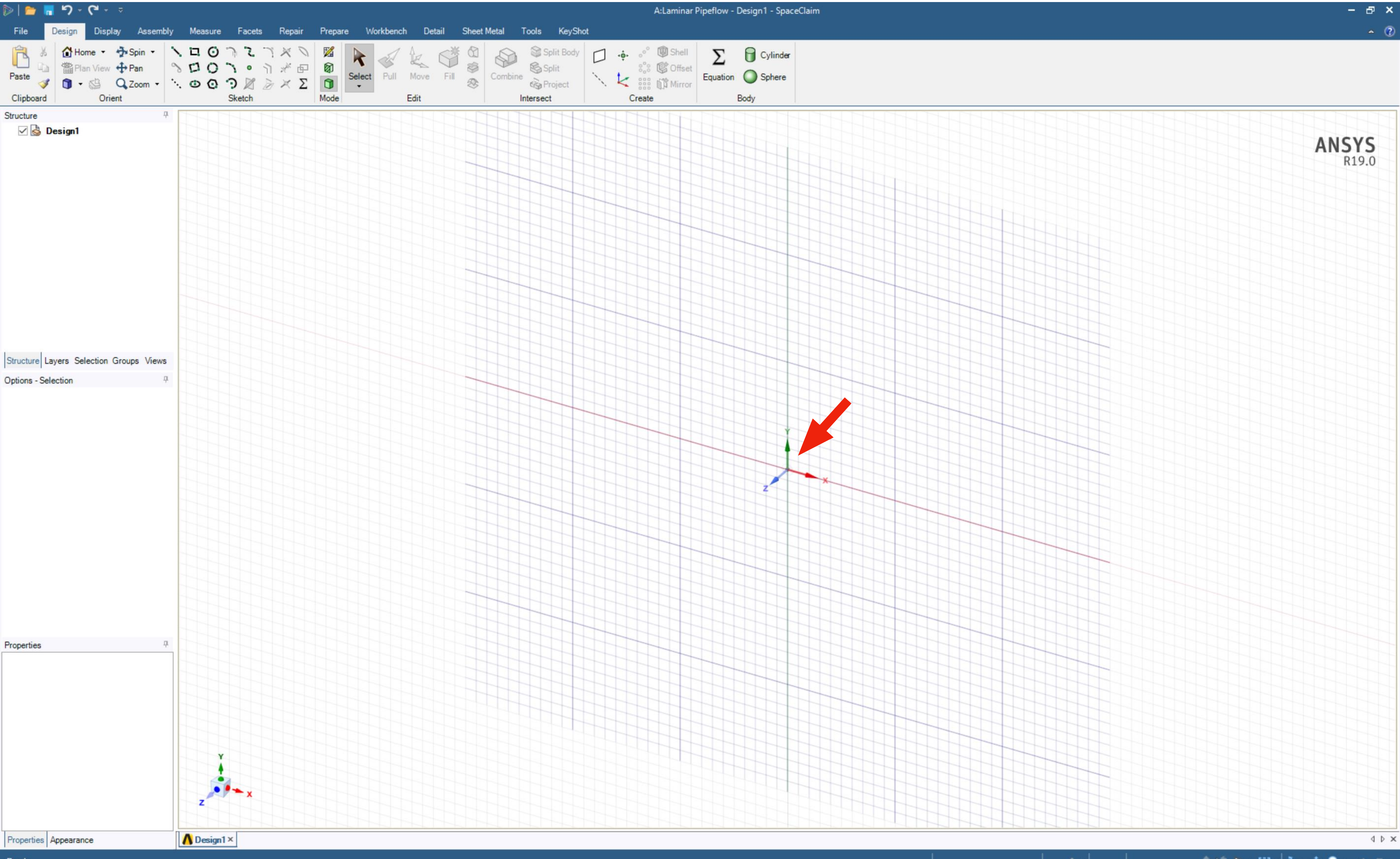
Modify Sketch Plane

Select New Sketch Plane



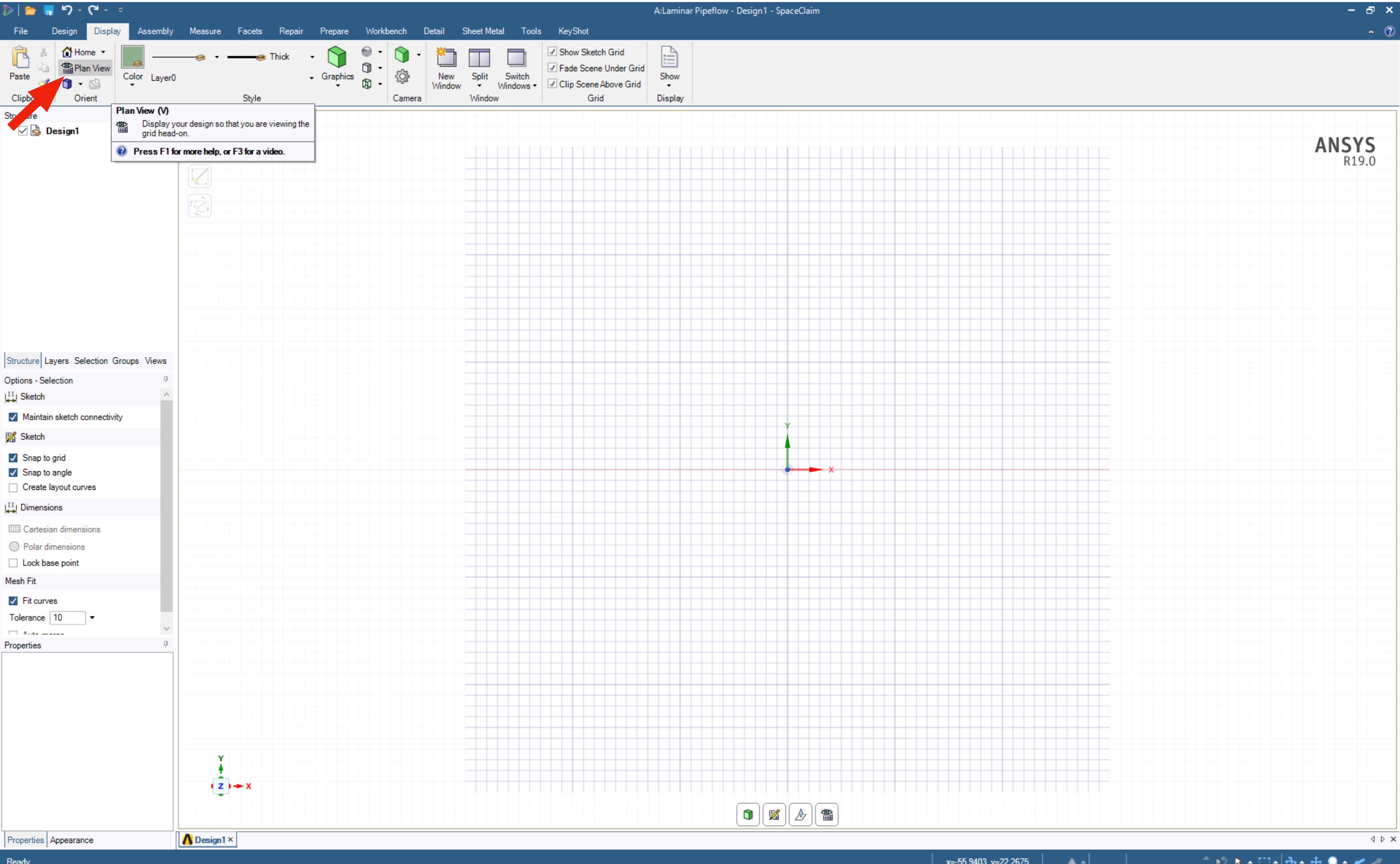
Modify Sketch Plane

Move the pointer on XY Plane / L-Click



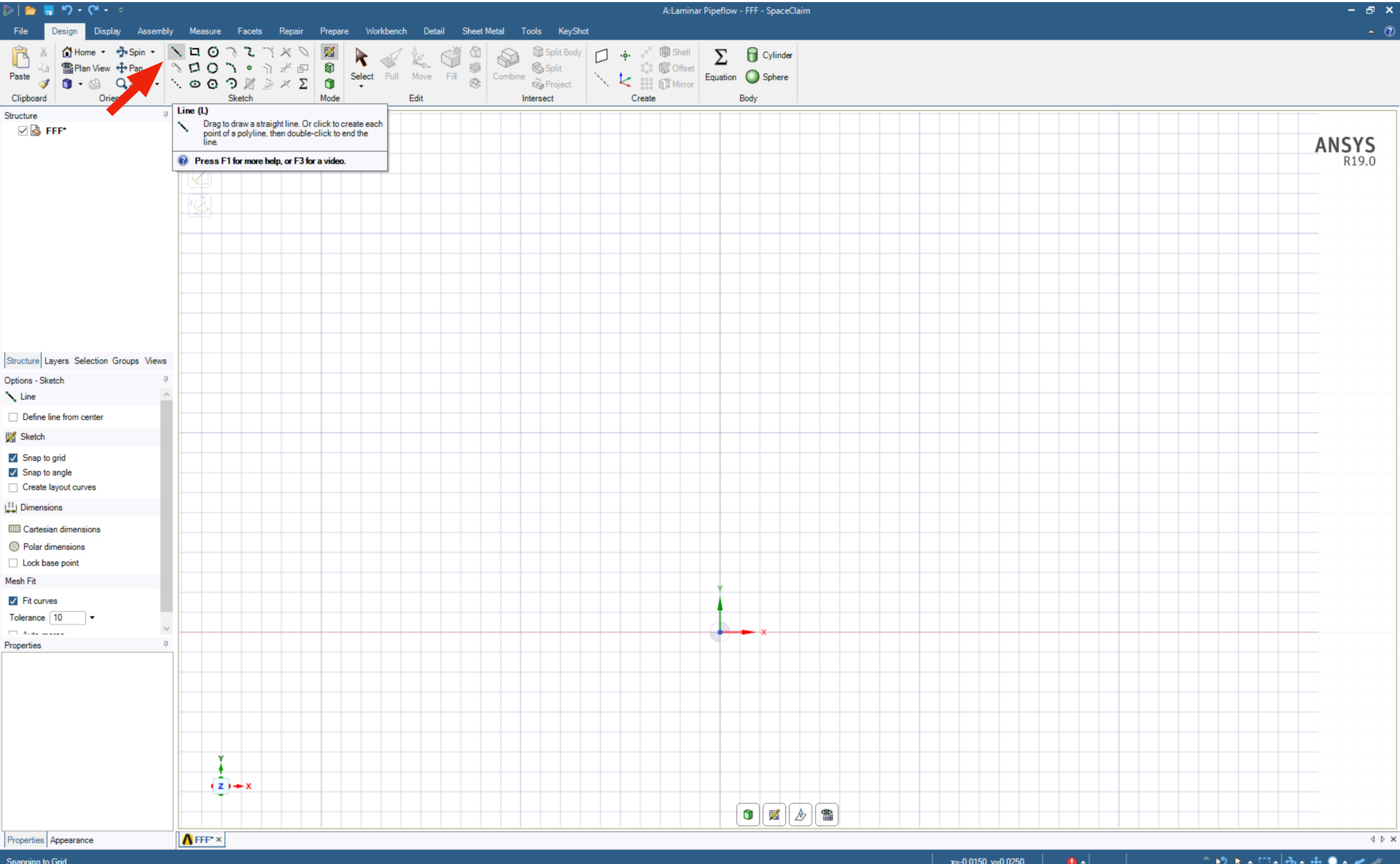
Plane View

Display/Plan View or (Ctrl-6)



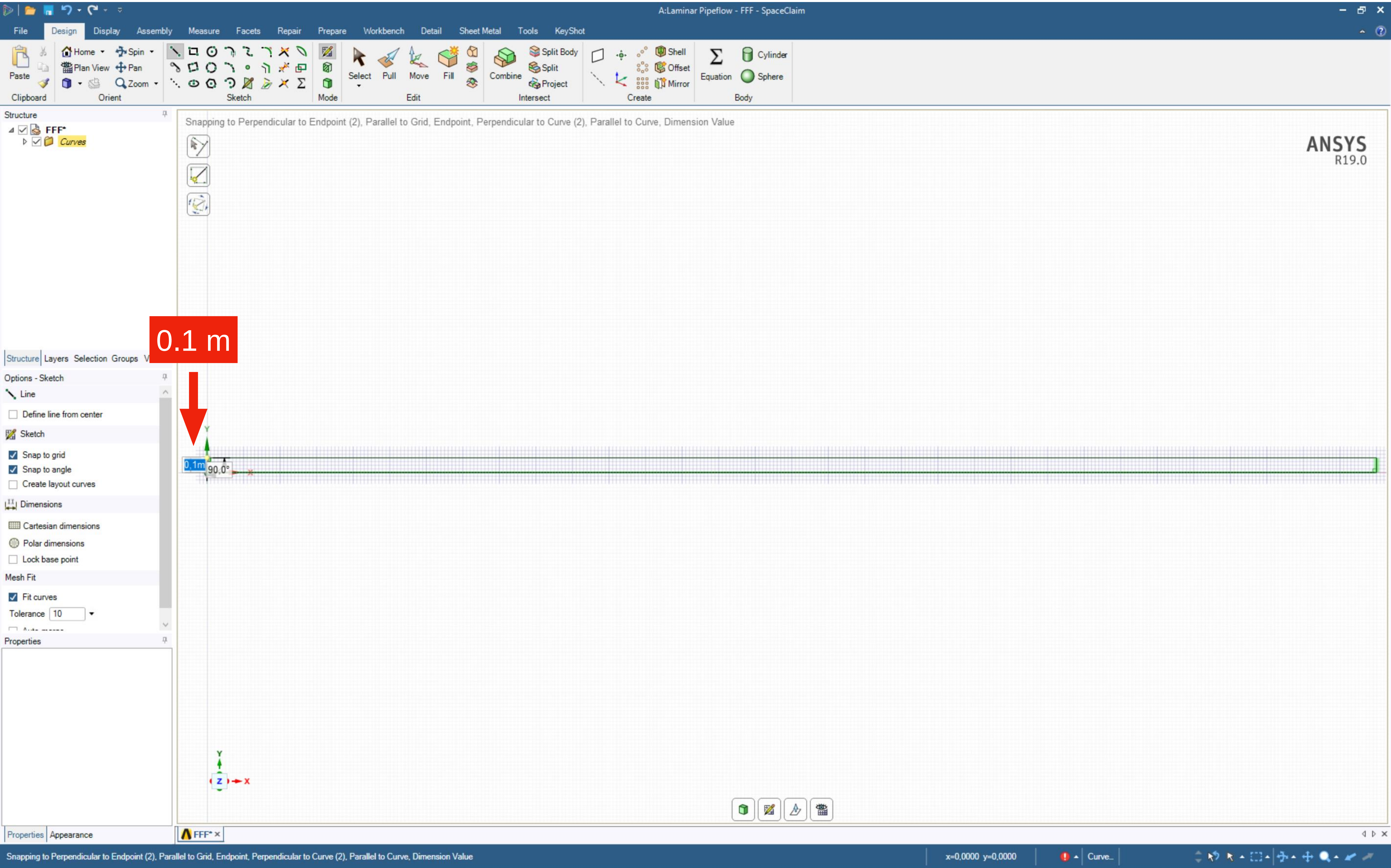
Draw the Cylinder Section Based on Given Dimensions

Design/Line icon



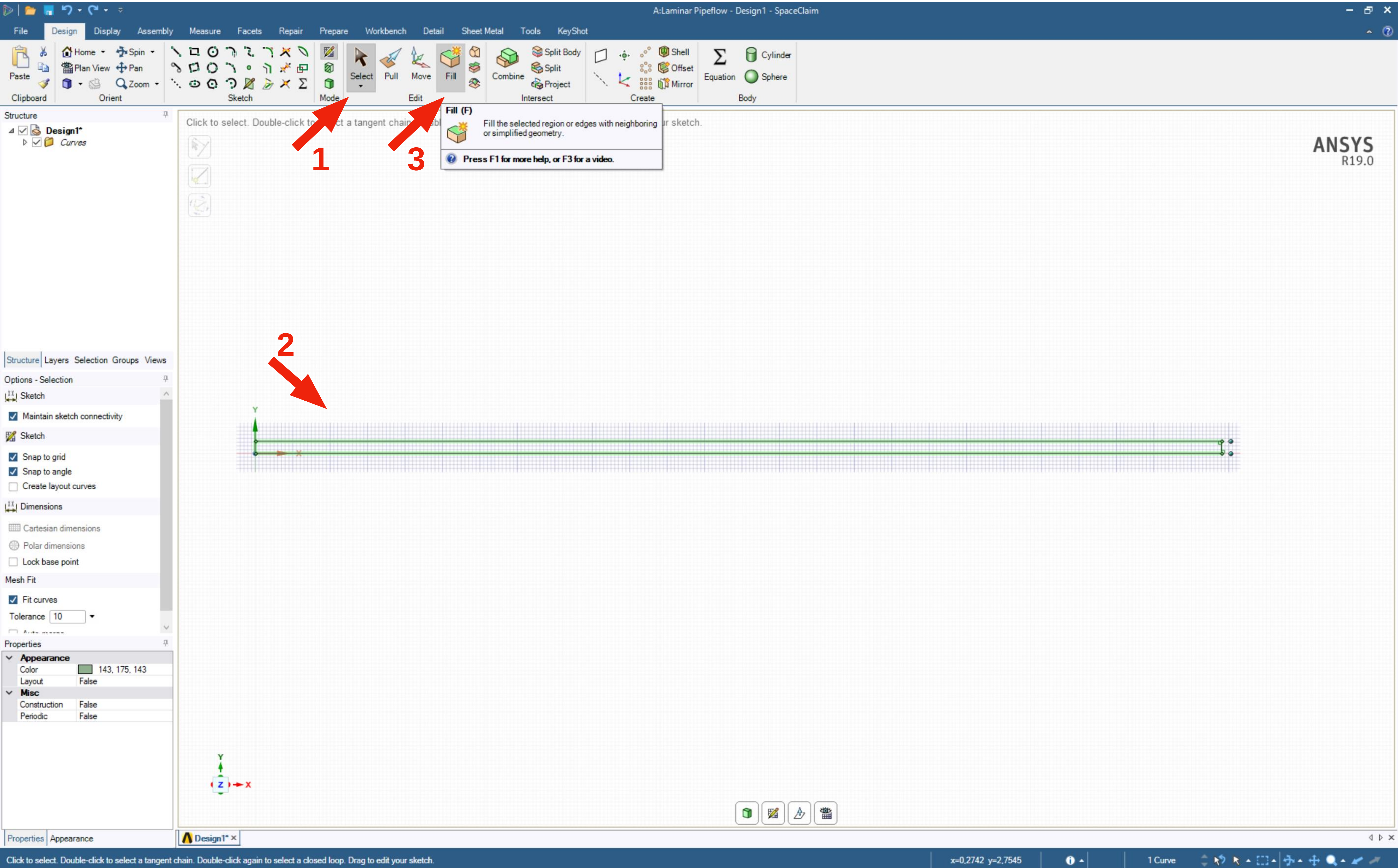
Draw the Cylinder Section Based on Given Dimensions

Be careful in drawing **straight** lines!

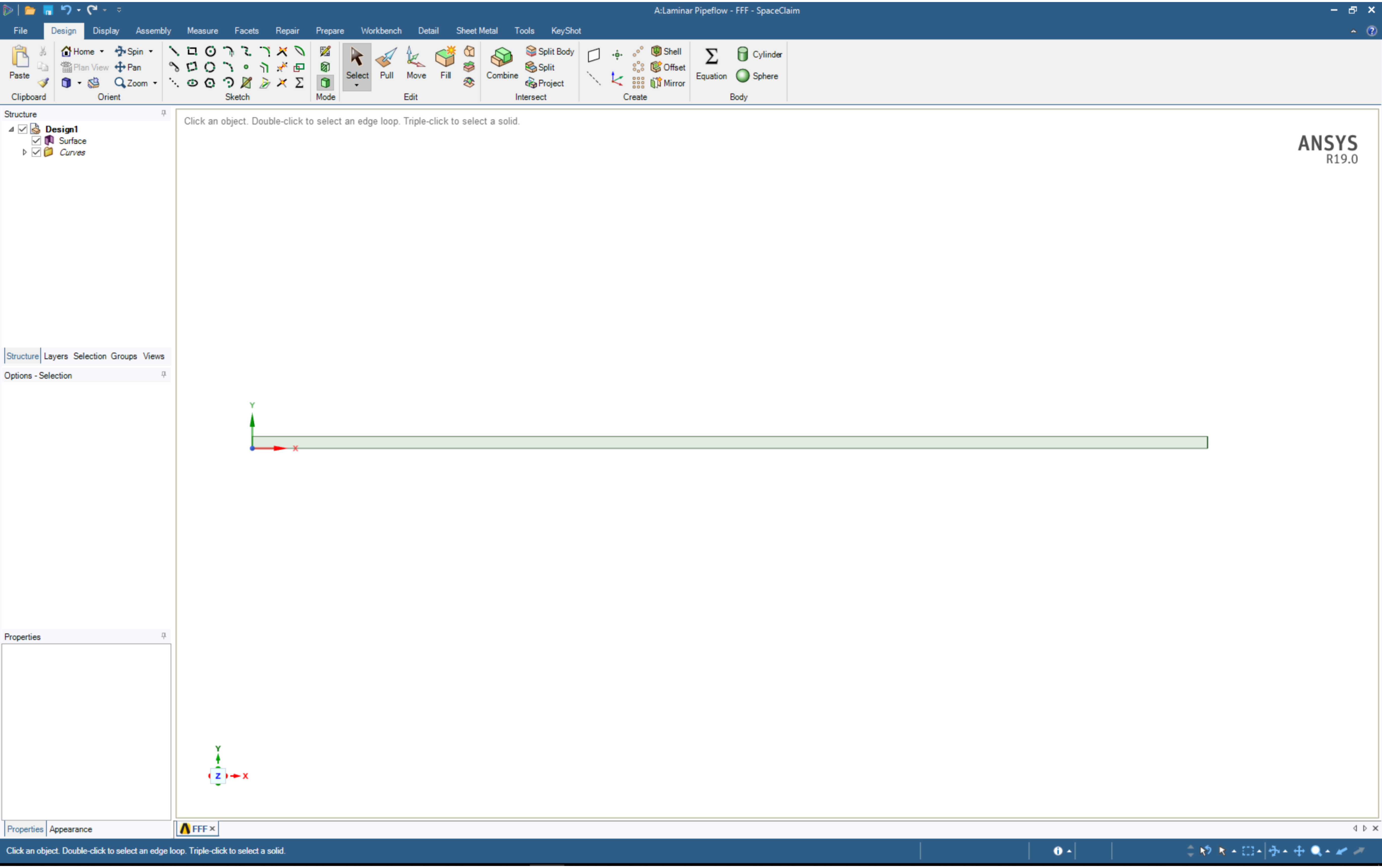


Fill the selected edges to create a surface

Design/select/select all the edges/Fill



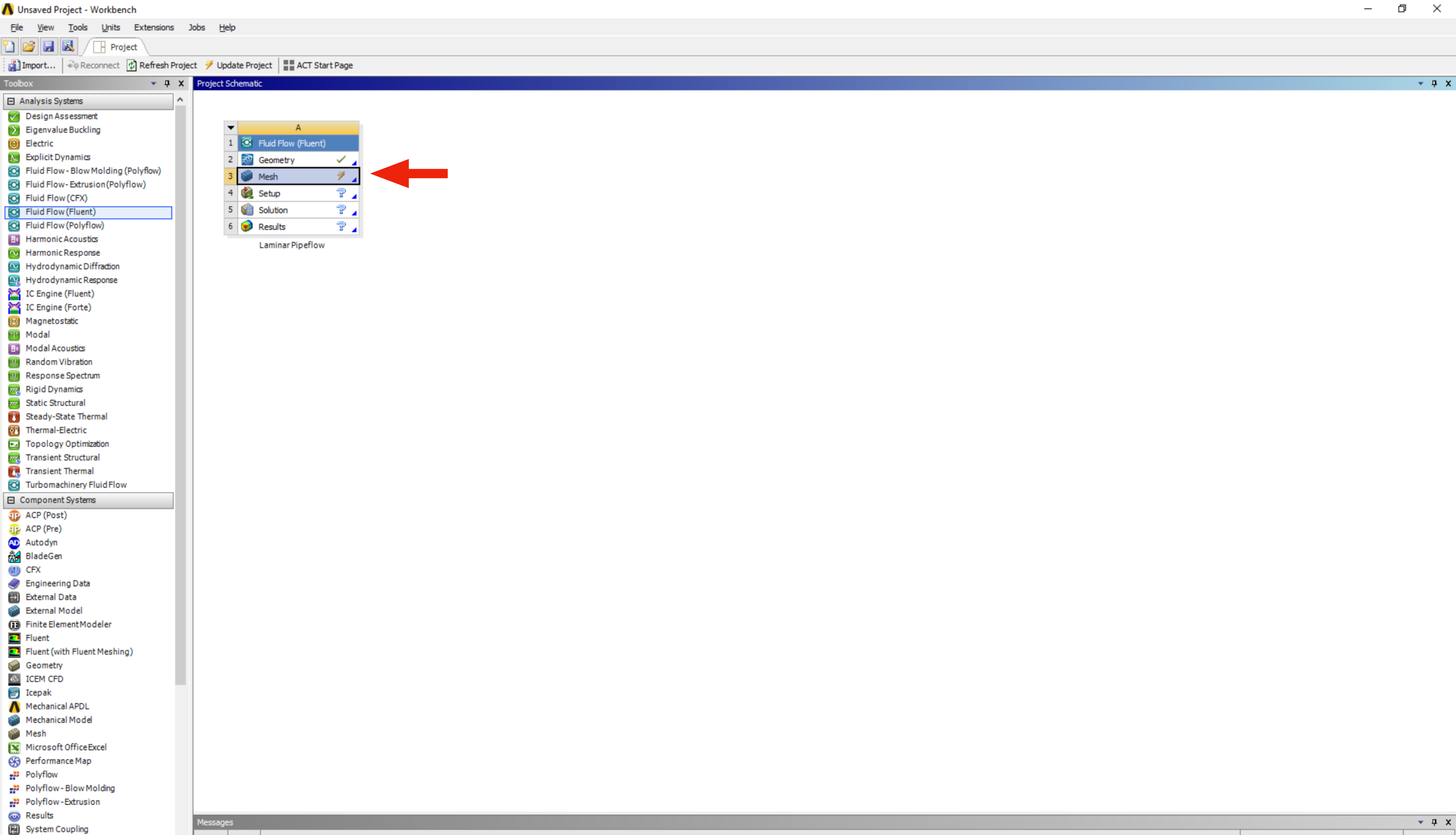
SpaceClaim Results



2- Run Mesh (From Workbench)

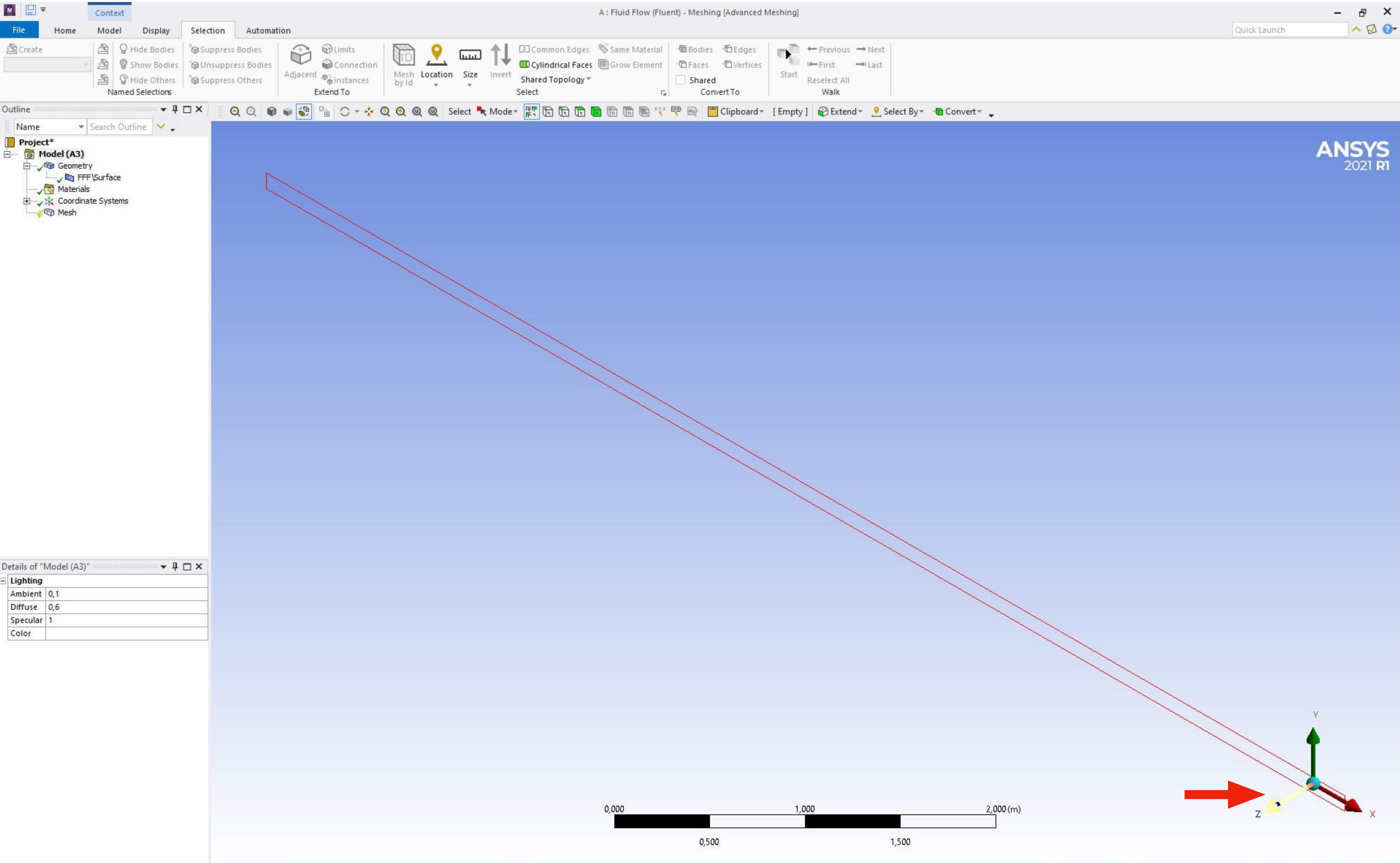
Control Green checkmark of Geometry/Save the project

Run Mesh from Workbench



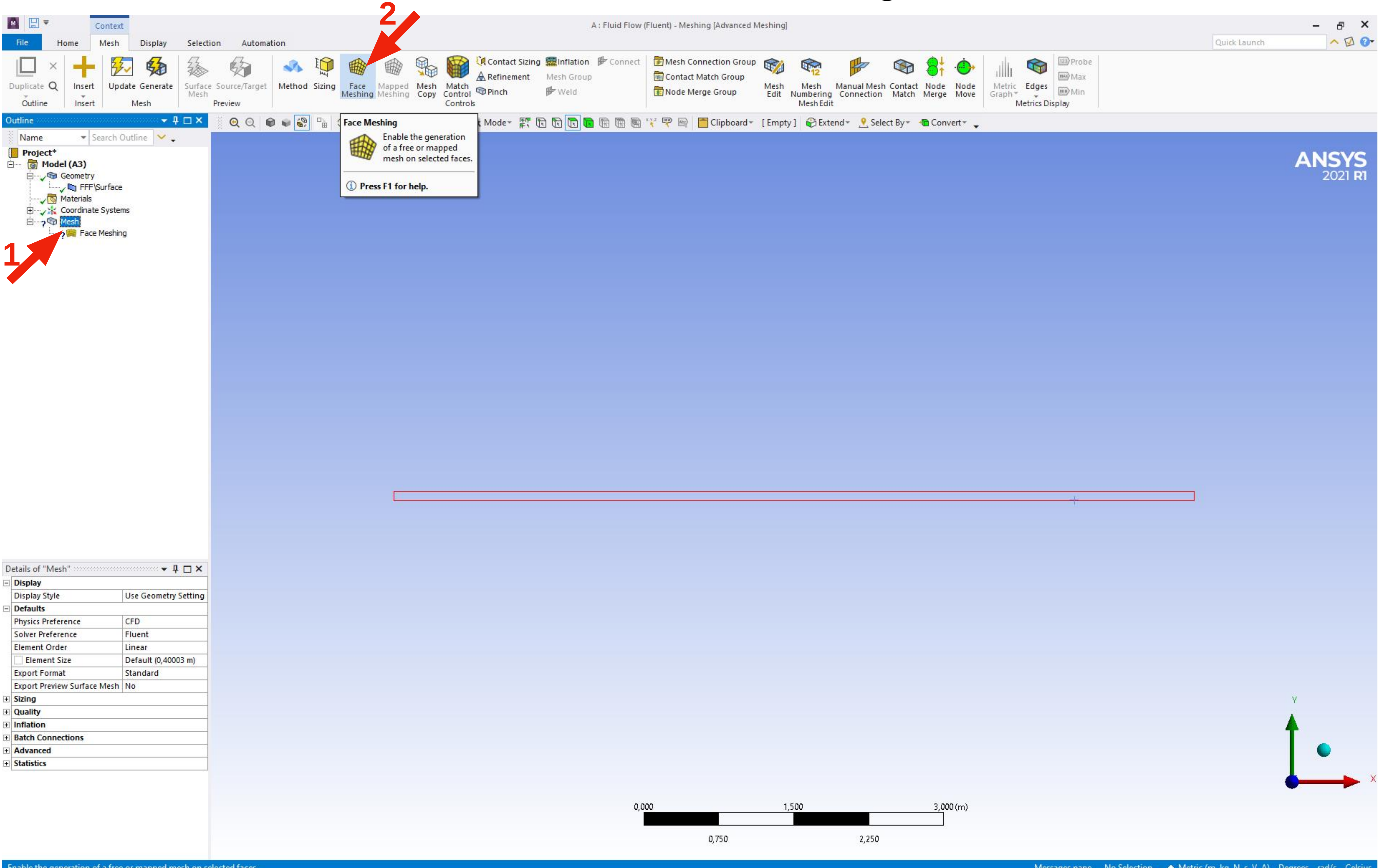
Plane View

L-Click on the z-axis



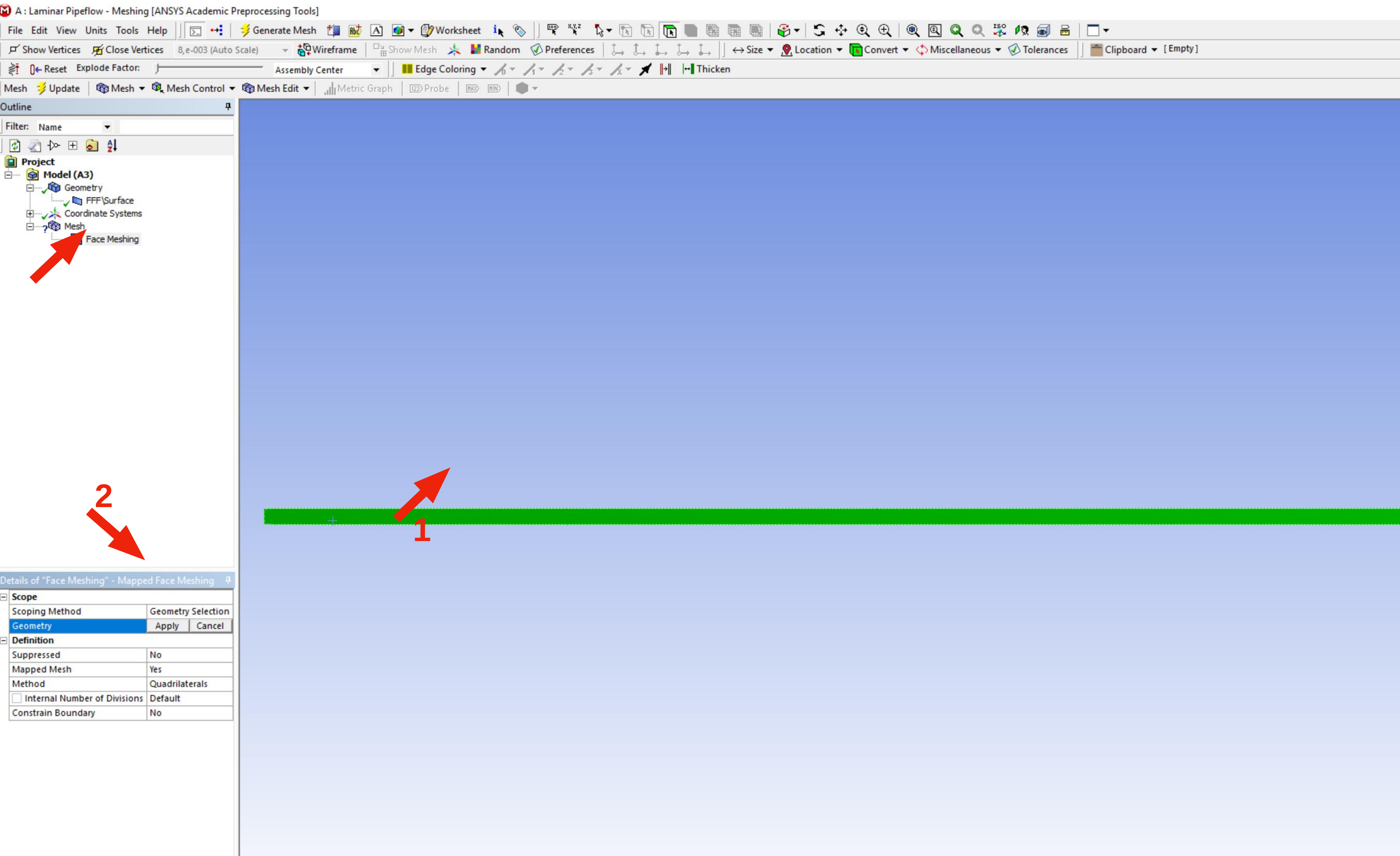
Face meshing

Click on Mesh at Outline window/Face Meshing



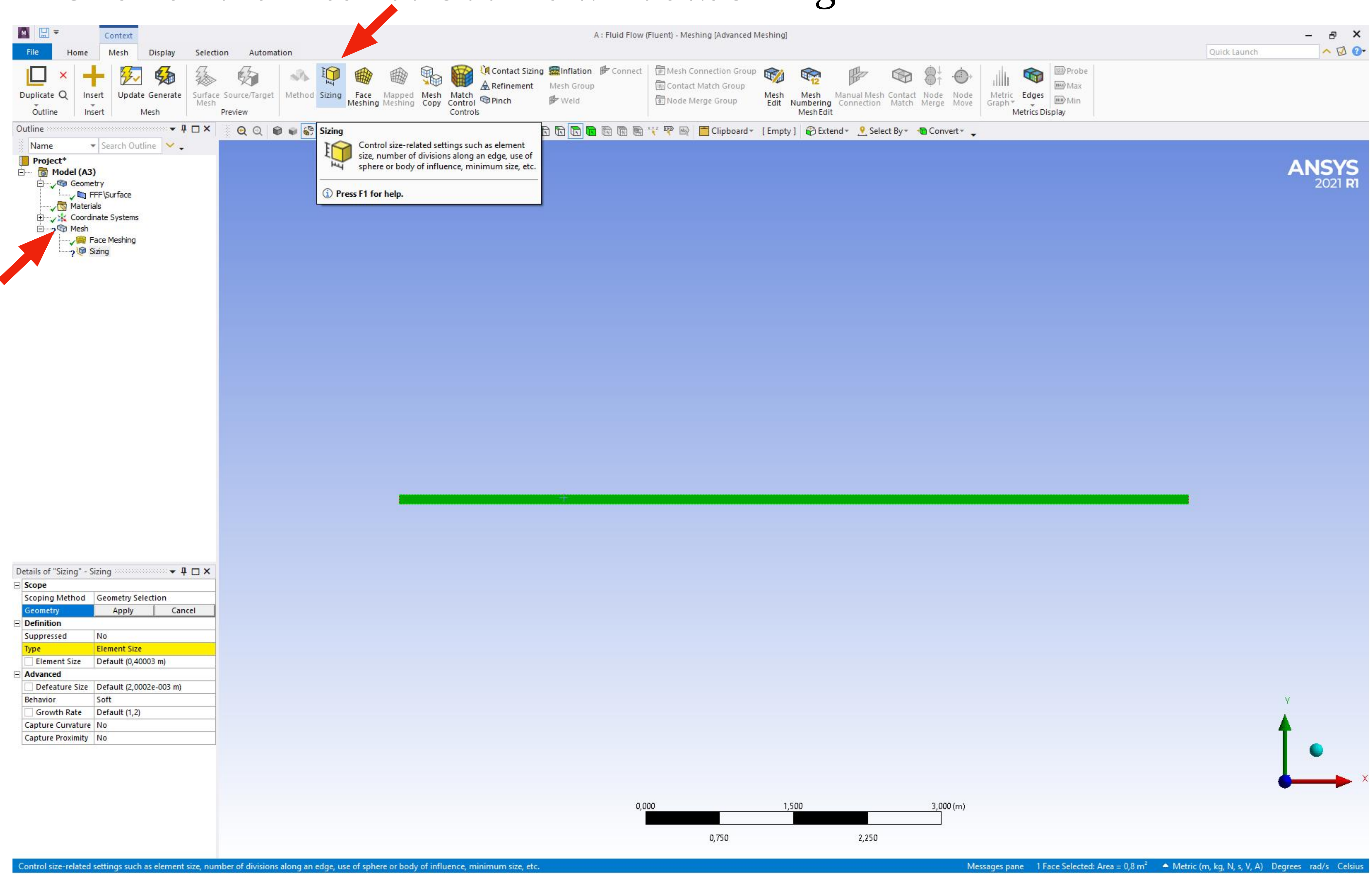
Face Meshing

Click on the created surface (to make it colourful)/click on Geometry/apply



Add Sizing

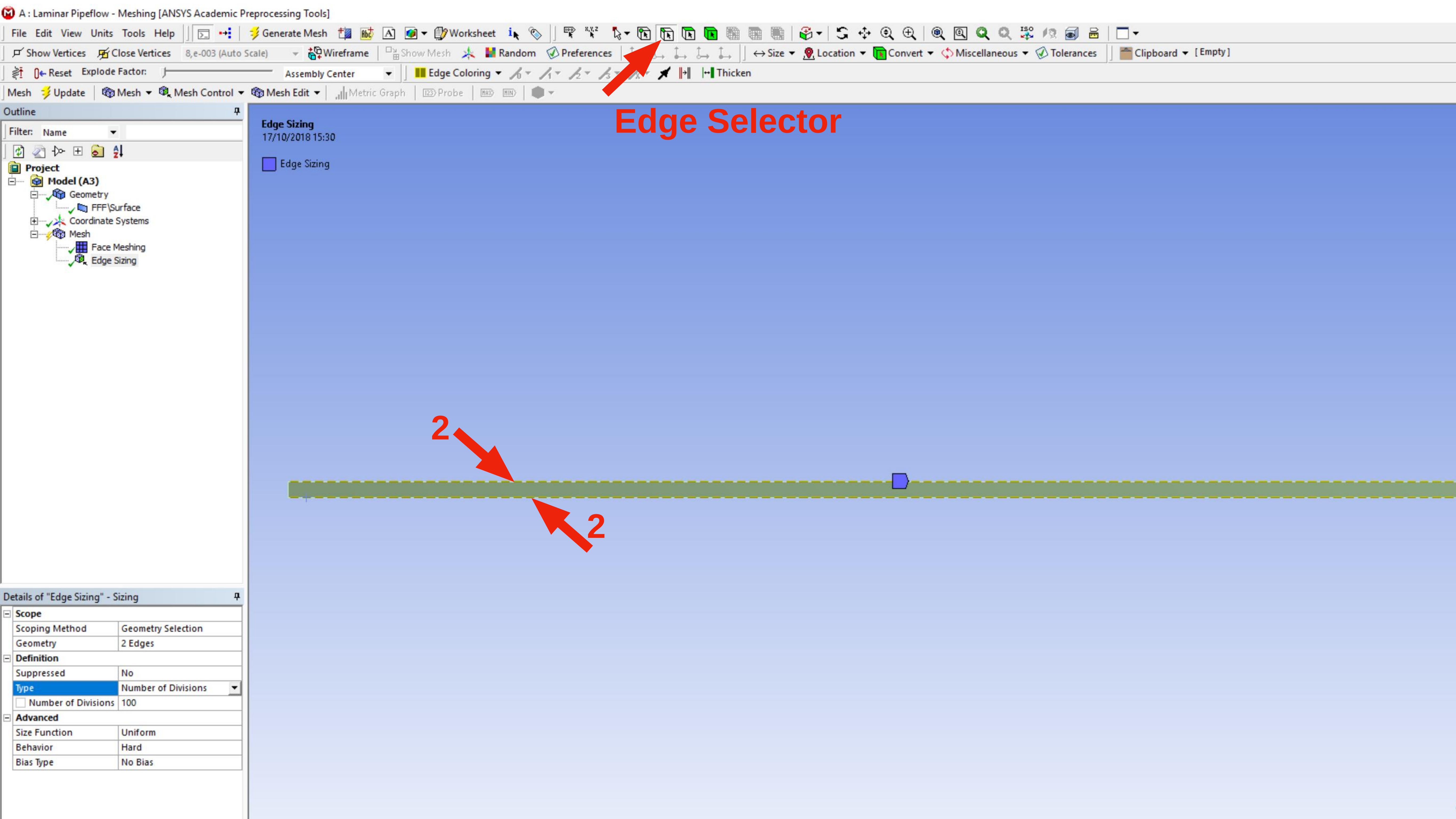
Click on the Mesh at Outline window/Sizing



2- Add Sizing

Click on Edge selector/Select both bigger edges/Apply

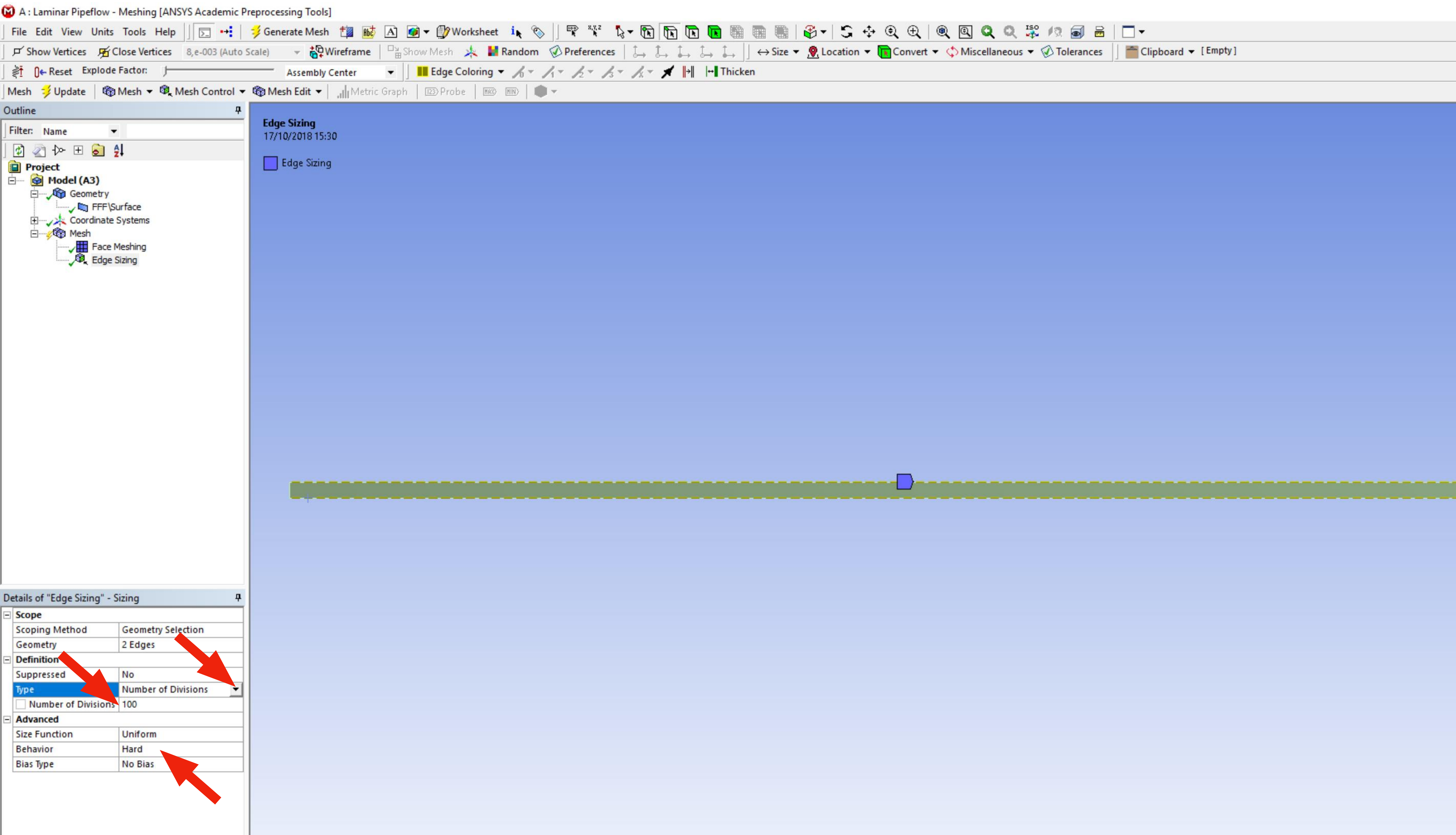
Note! Use Ctrl key to add an edge/



Sizing: Define Number of Divisions

Set Type to Number of Divisions/Set **100** Divisions

Set Behavior to **Hard**



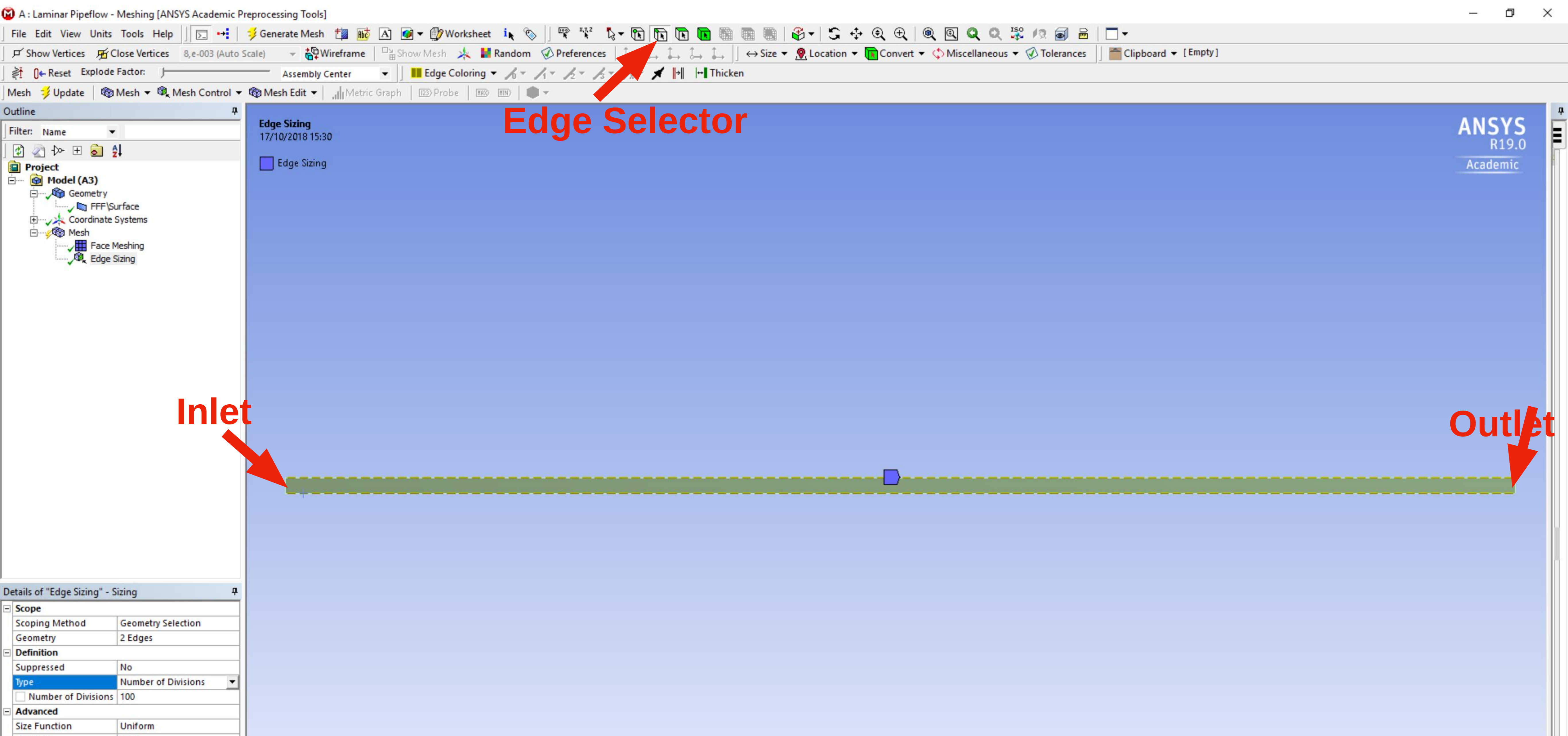
Add Sizing to Inlet and Outlet

Click on the Mesh at Outline window/Sizing

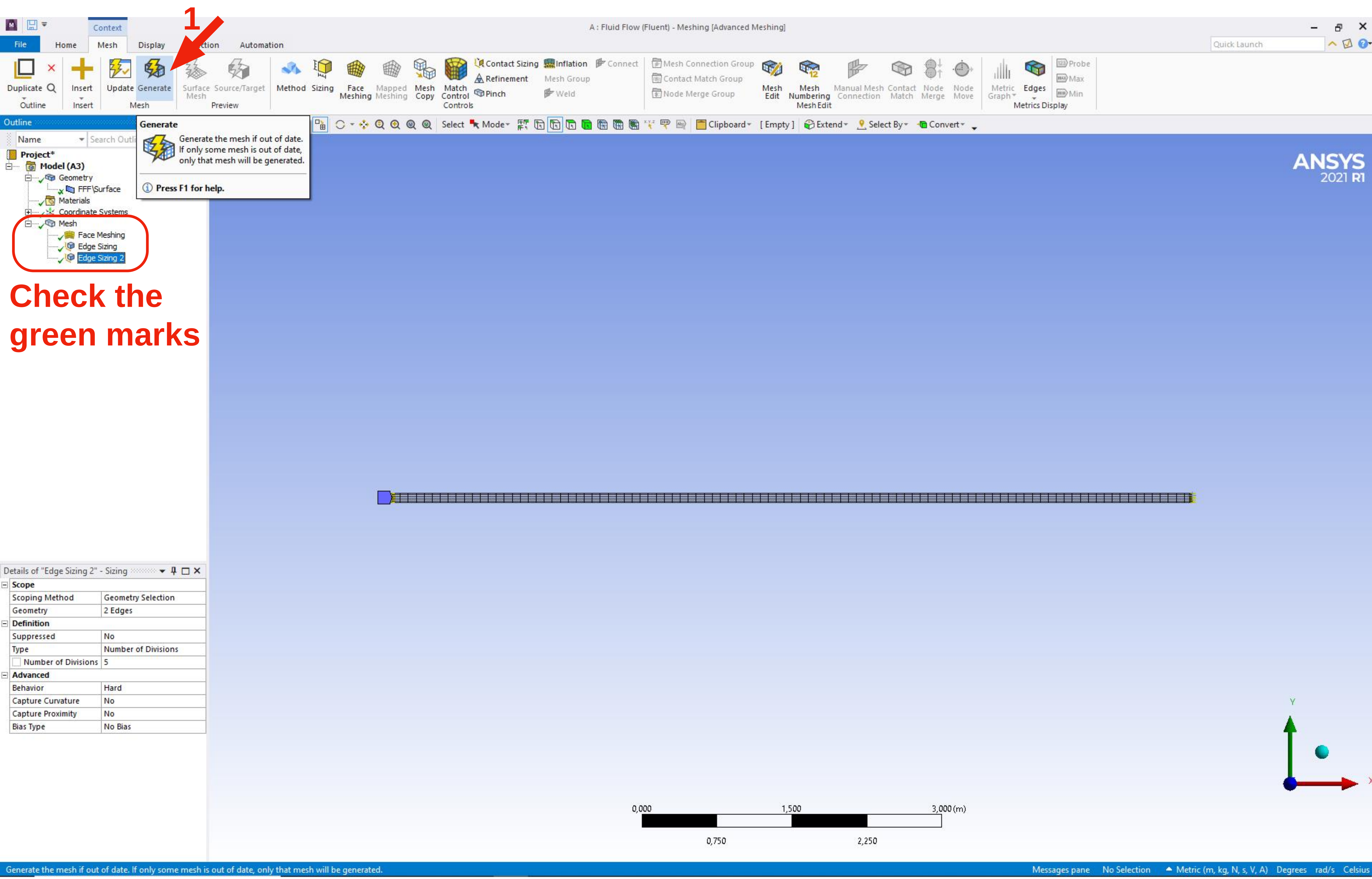
Click on Edge selector/Select the **inlet** and **outlet** edges/Apply

Set Type to Number of Divisions/Set **5** Divisions

Set Behavior to **Hard**

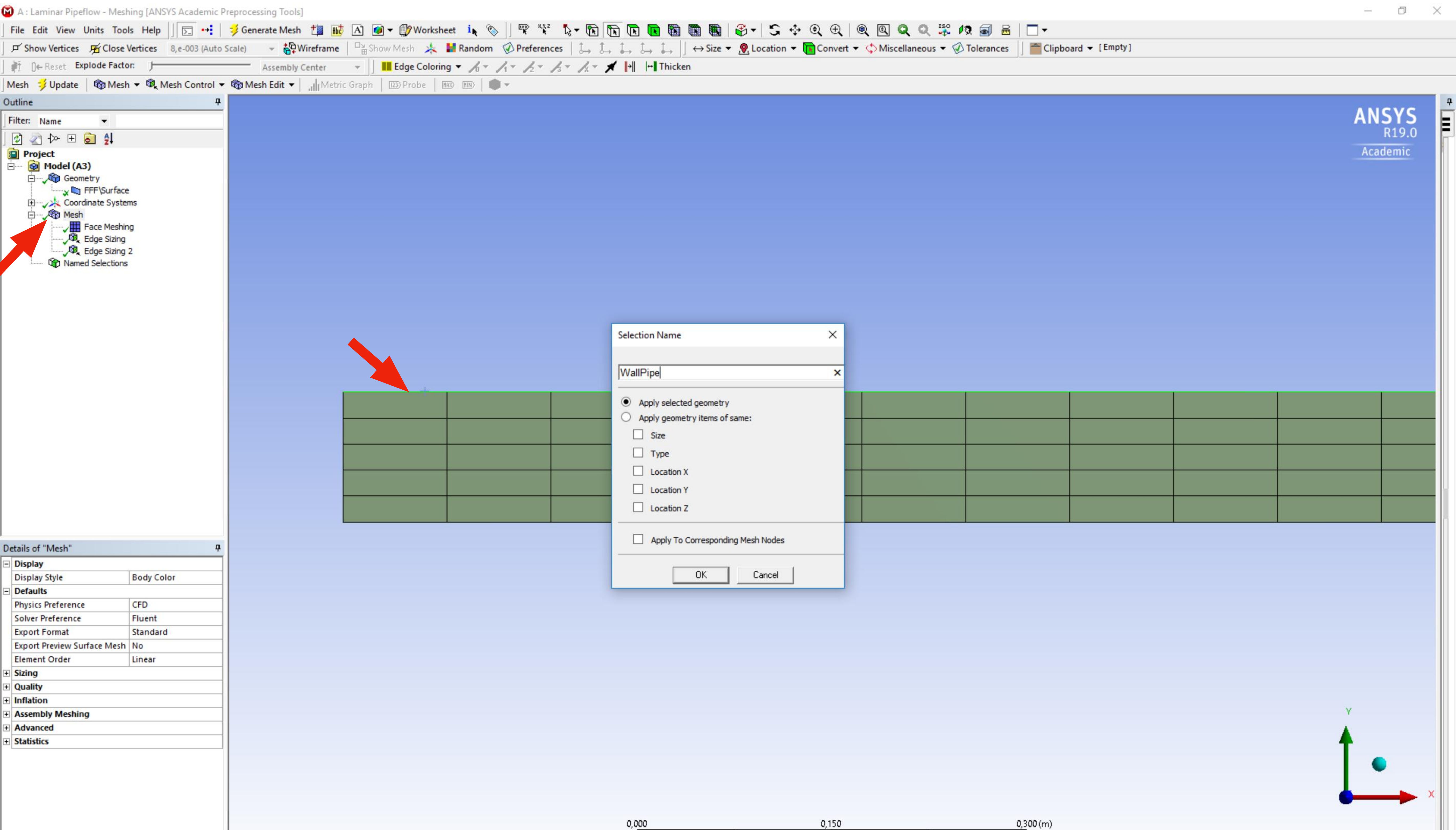


Generate Mesh



Create Named Selection

Click on Edge selector/Select the upper edge/R-Click on the edge /Create Named Selection/Enter the name “WallPipe”



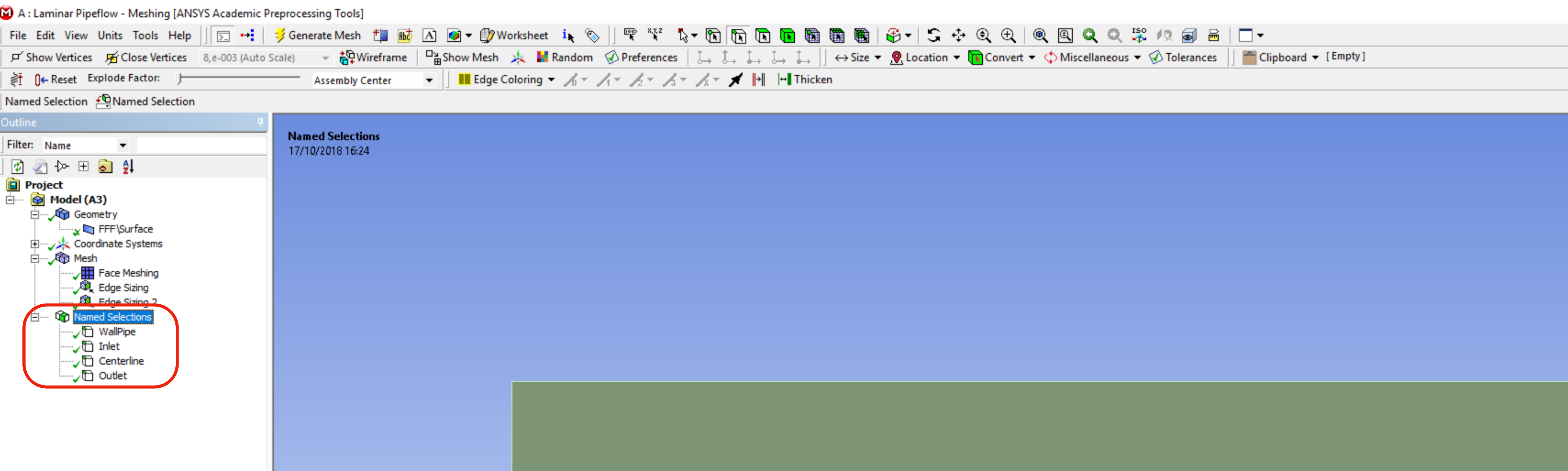
Create Named Selection for other edges

Click on the Edge selector/Select the lower edge/R-Click on the edge /Create Named Selection/Enter the name “Centerline”

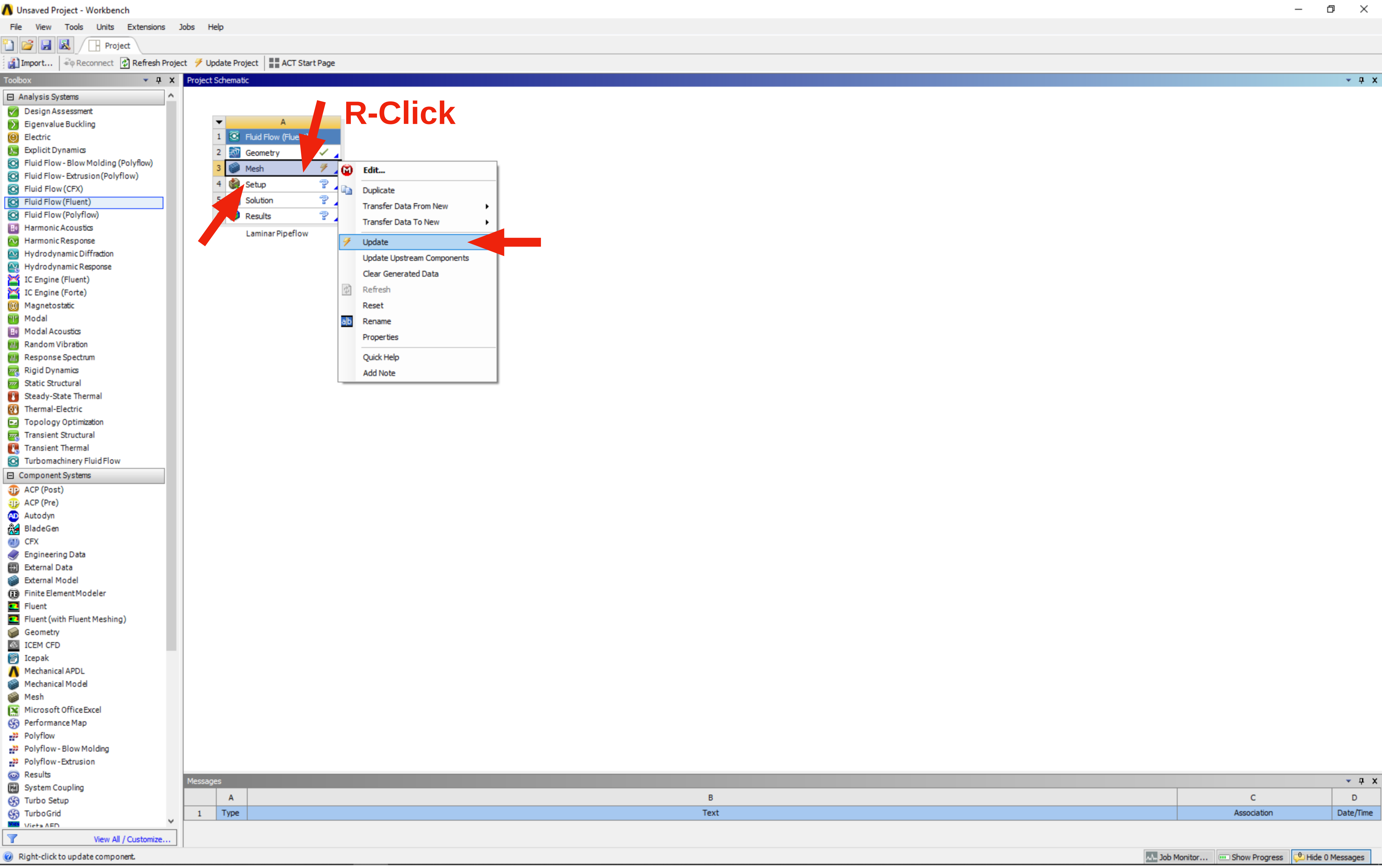
Click on the Edge selector/Select the inlet edge/R-Click on the edge /Create Named Selection/Enter the name “Inlet”

Click on the Edge selector/Select the outlet edge/R-Click on the edge /Create Named Selection/Enter the name “Outlet”

Control the Named Selections by clicking the names in the list

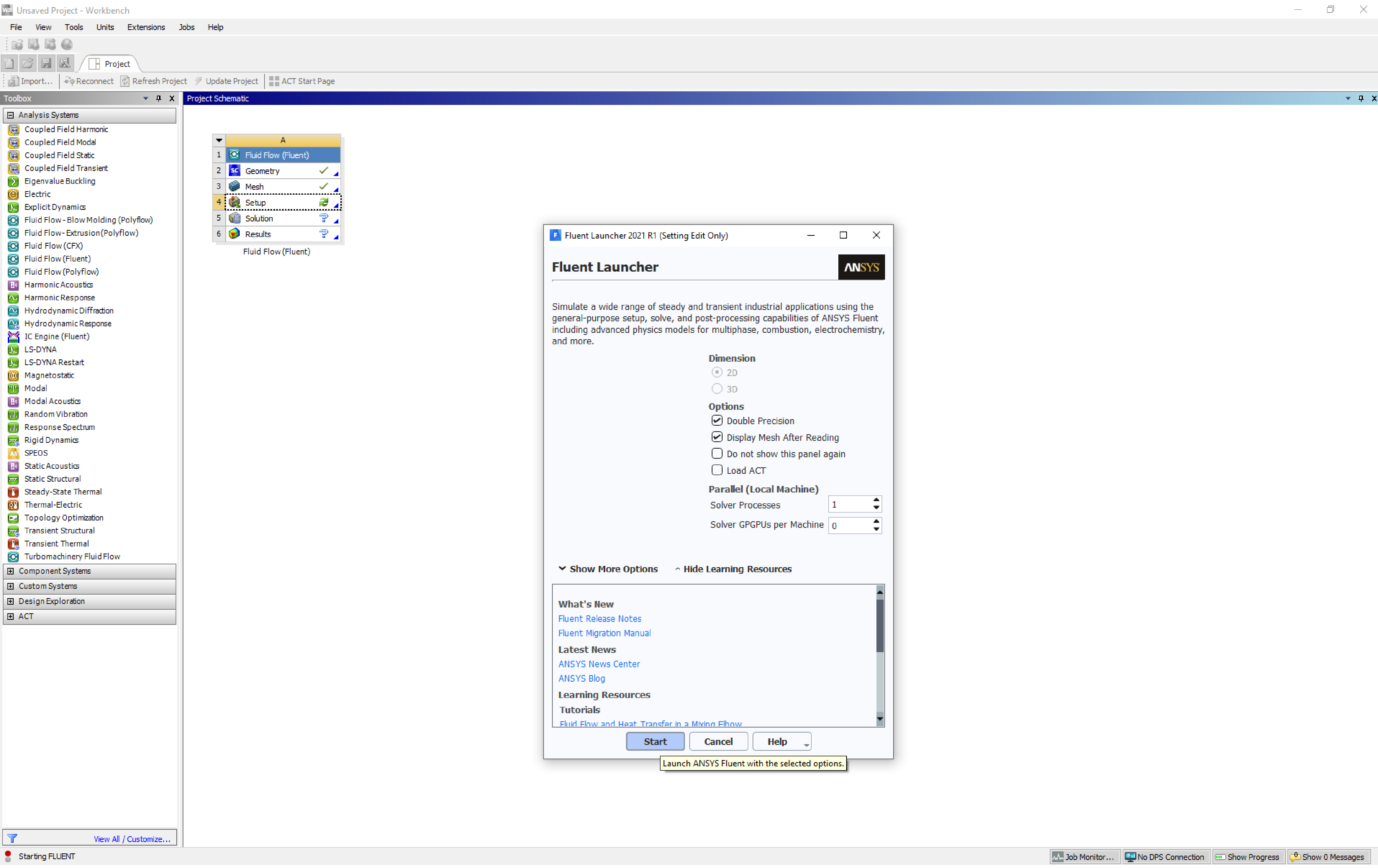


Update Meshing at Workbench and Run Setup (D-Click)



3- Fluent Launcher

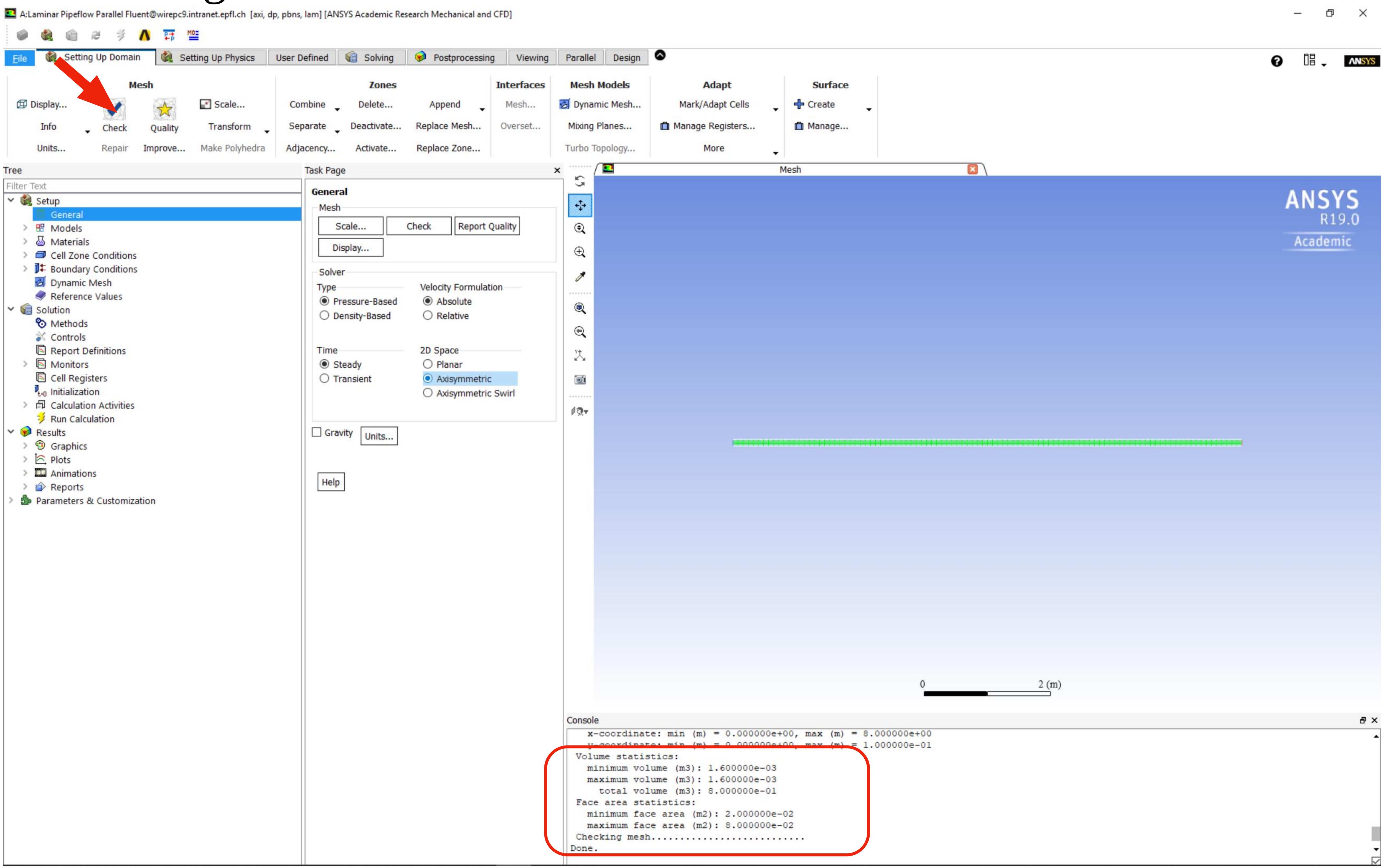
Click on the Double Precision icon



Check Mesh

Click on check/perform mesh check

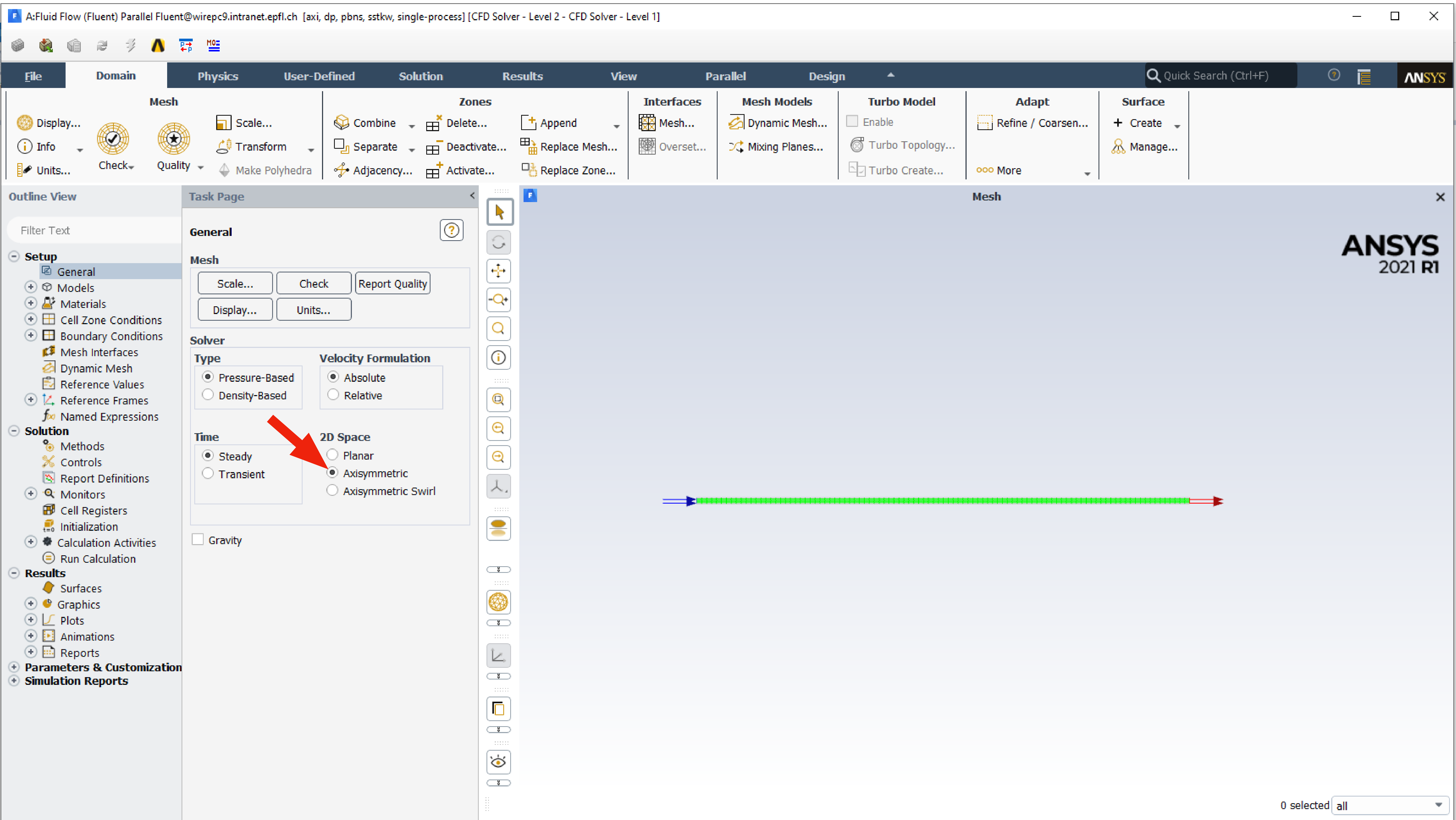
Check the given information



Setup/General

At the Tree Menu double click on **General**

Click on **Axisymmetric** Checkbox

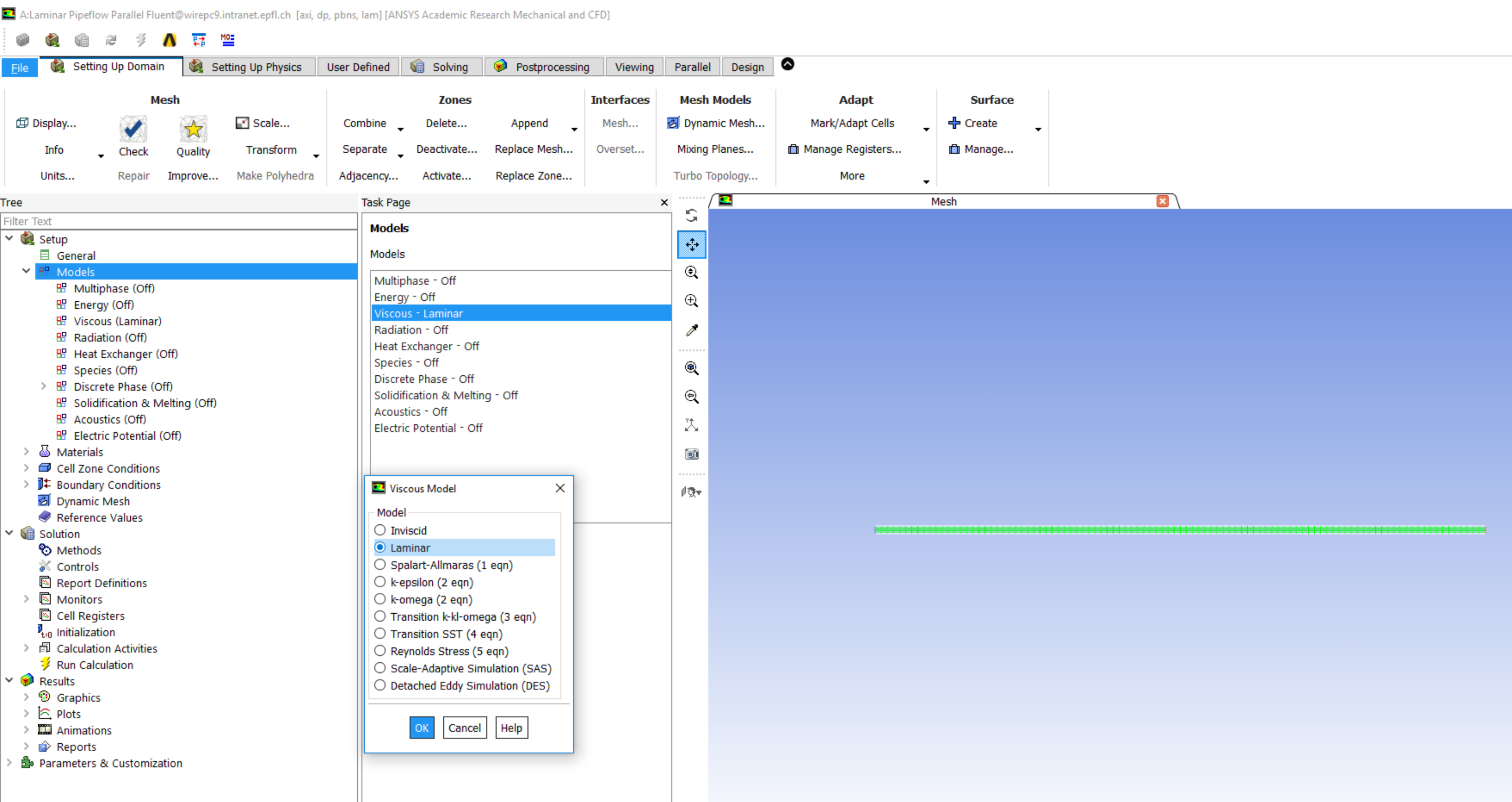


Setup/Models

At the Tree Menu double click on **Models**

Double Click on Viscous - Choose Laminar

Select Laminar and OK



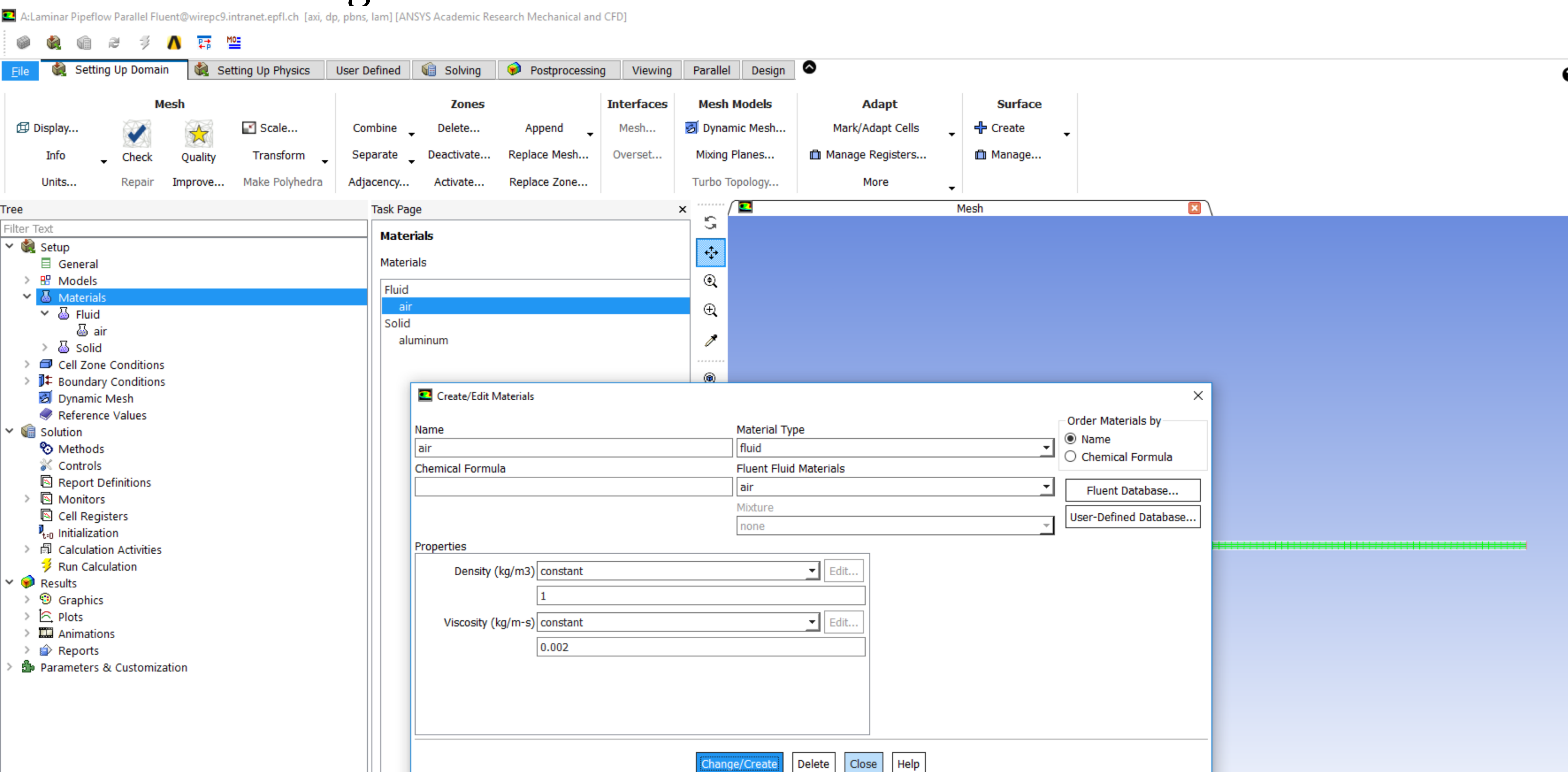
Setup/Materials

At the Tree Menu double click on **Materials**

Double Click on **air** (Fluid)

Change Density to **1** and Viscosity to **2e-3**

Click on Change/Create then Close

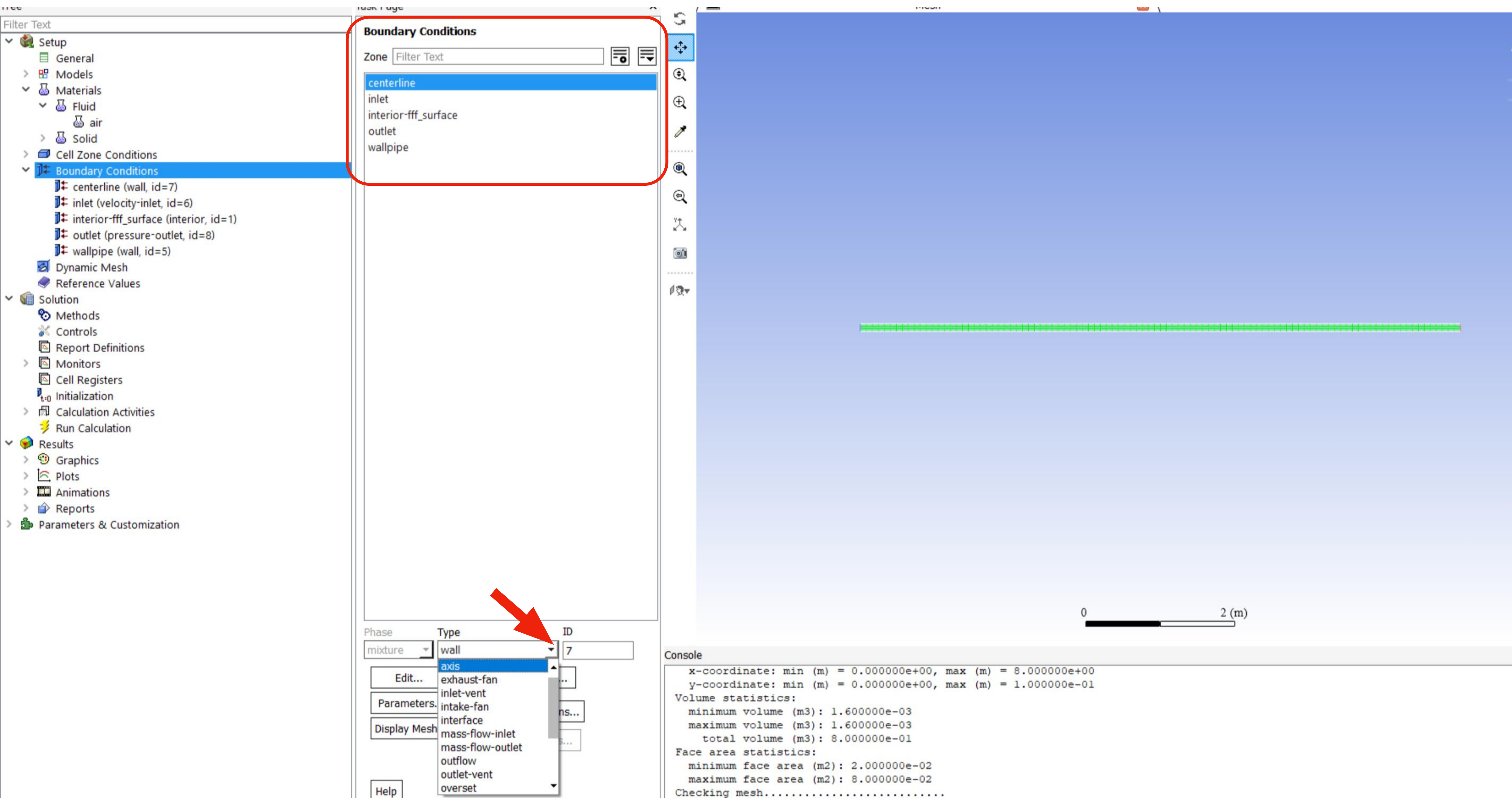


Setup/Boundary Conditions

At the Tree Menu double click on **Boundary Conditions**

Left Click on **Centerline**

Change Type to **Axis** then OK



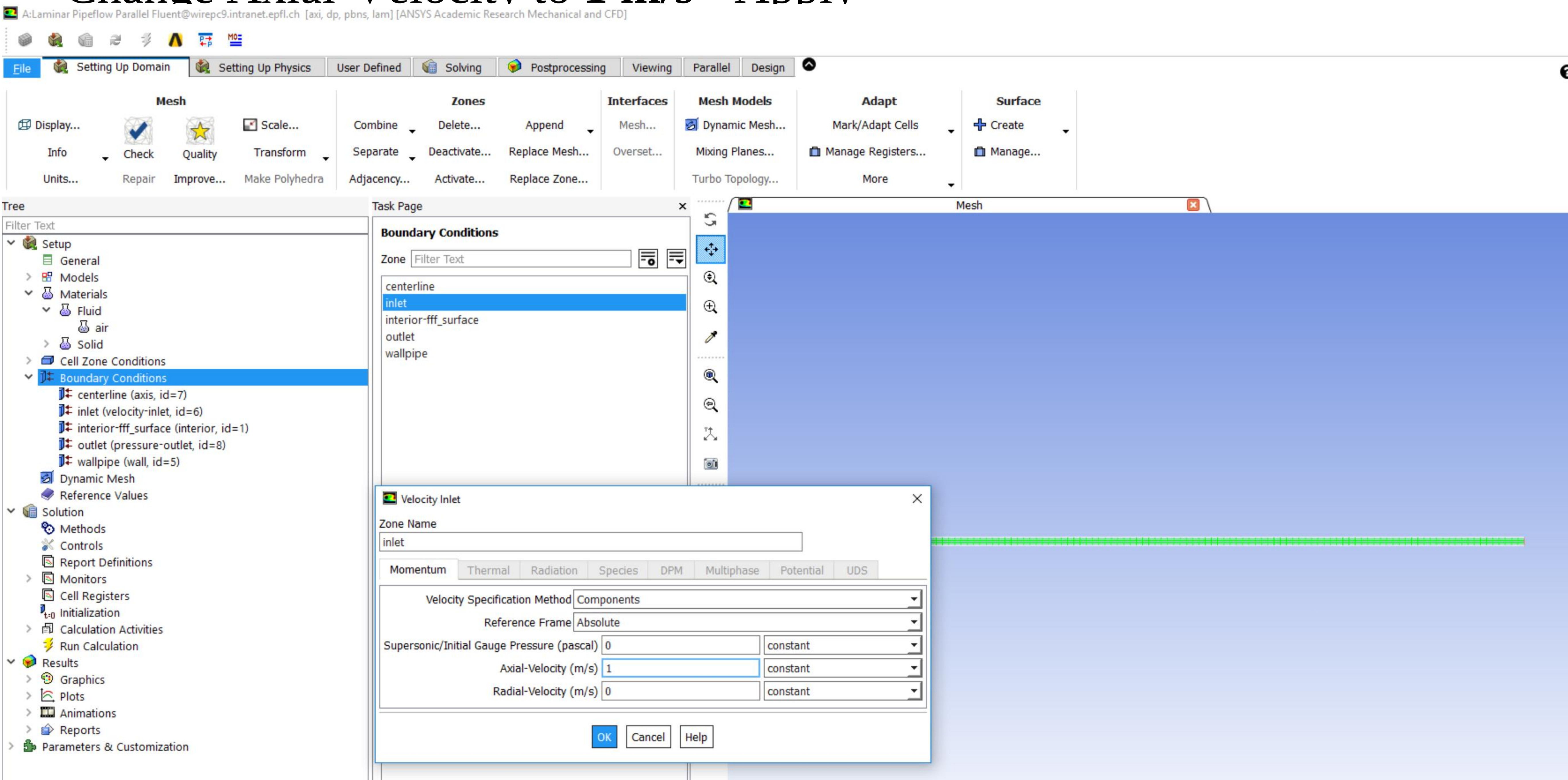
Setup/Boundary Conditions

At the Tree Menu double click on **Boundary Conditions**

Double Click on **Inlet**

Change Velocity Specification Method to **Components**

Change Axial-Velocity to **1 m/s** - Apply

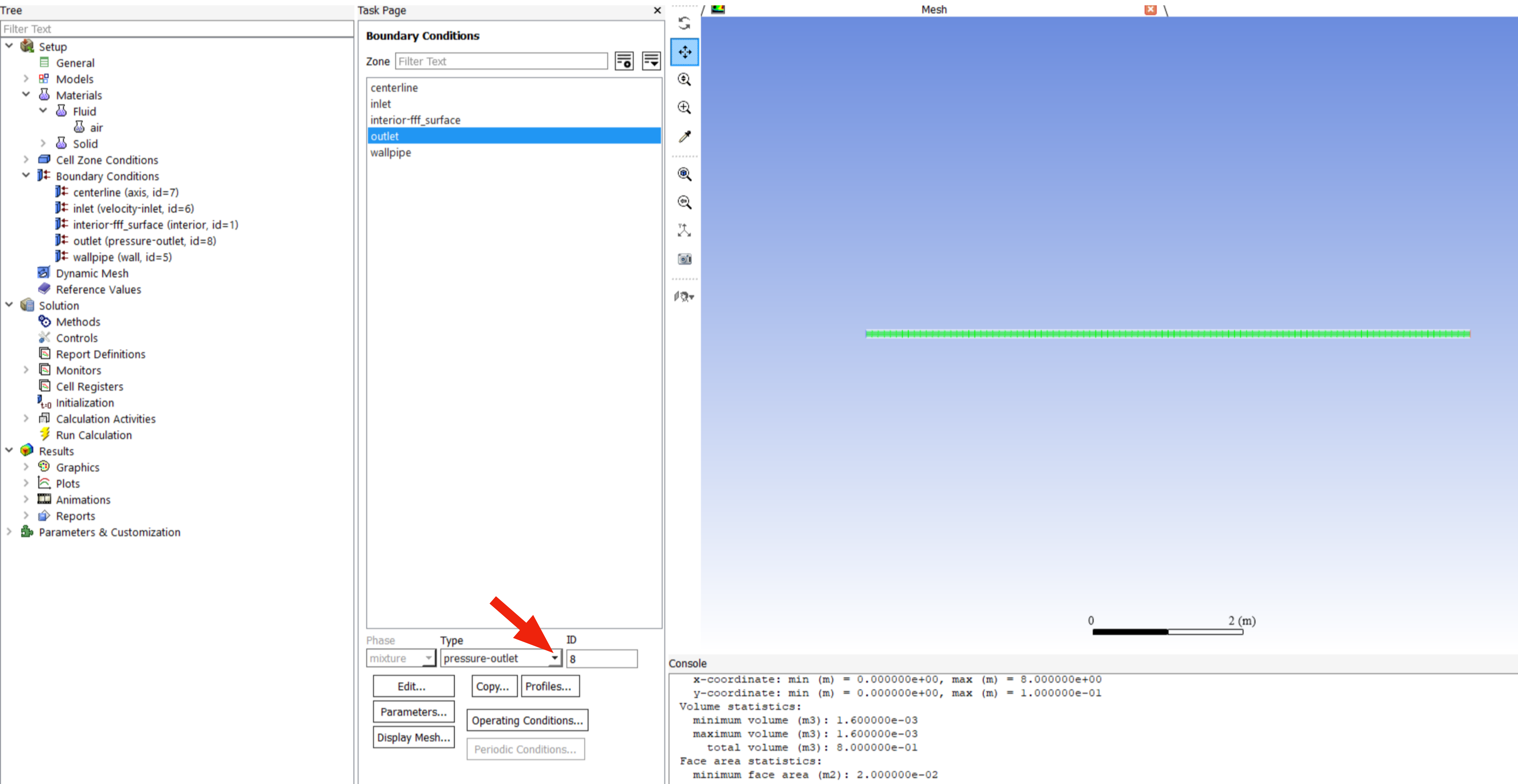


Setup/Boundary Conditions

At the Tree Menu double click on **Boundary Conditions**

Left Click on **Outlet**

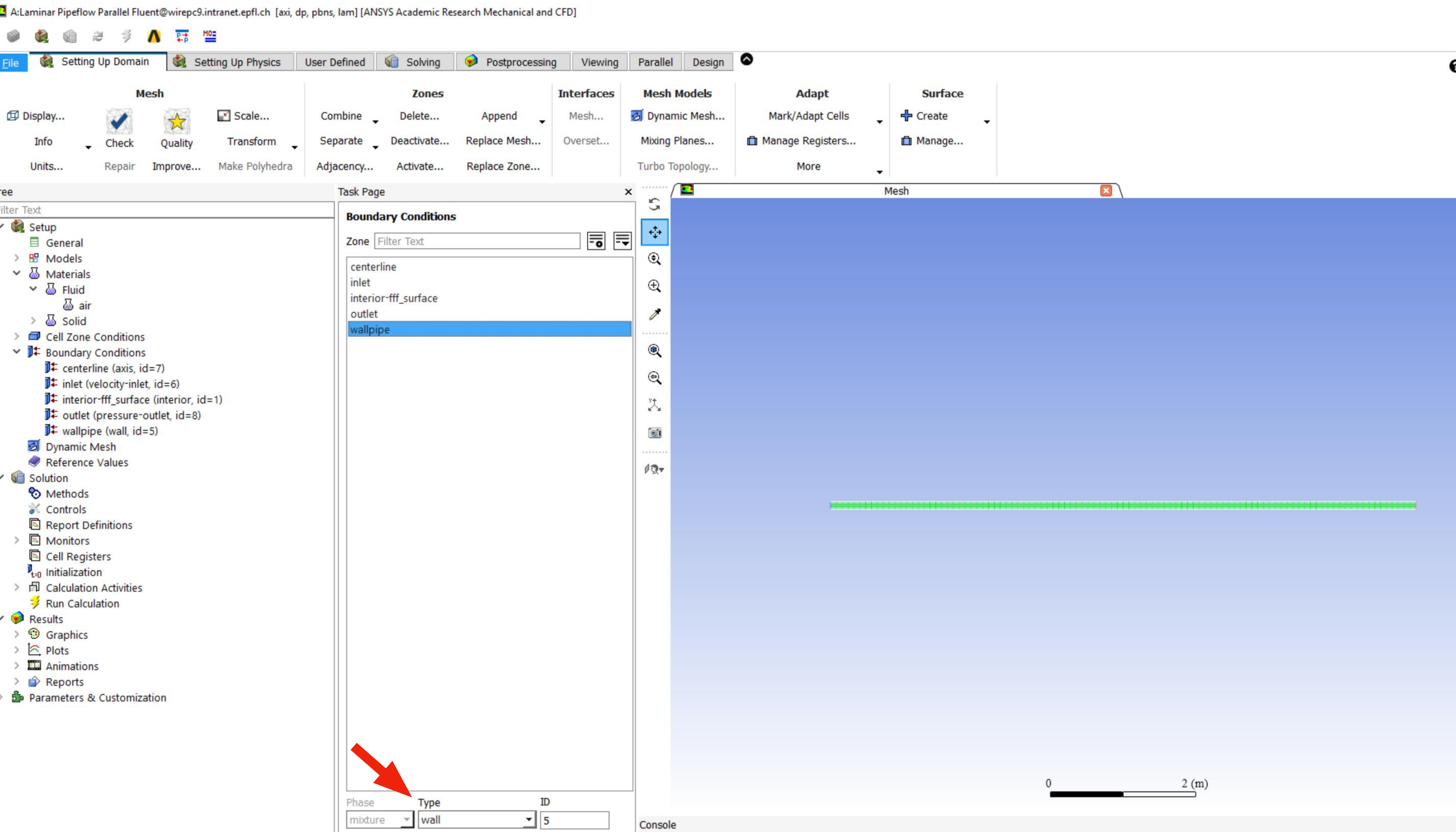
Check if the Type is **Pressure-outlet**



Setup/Boundary Conditions

At the Tree Menu double click on **Boundary Conditions**

Left Click on **WallPipe -** Check if the Type is **Wall**

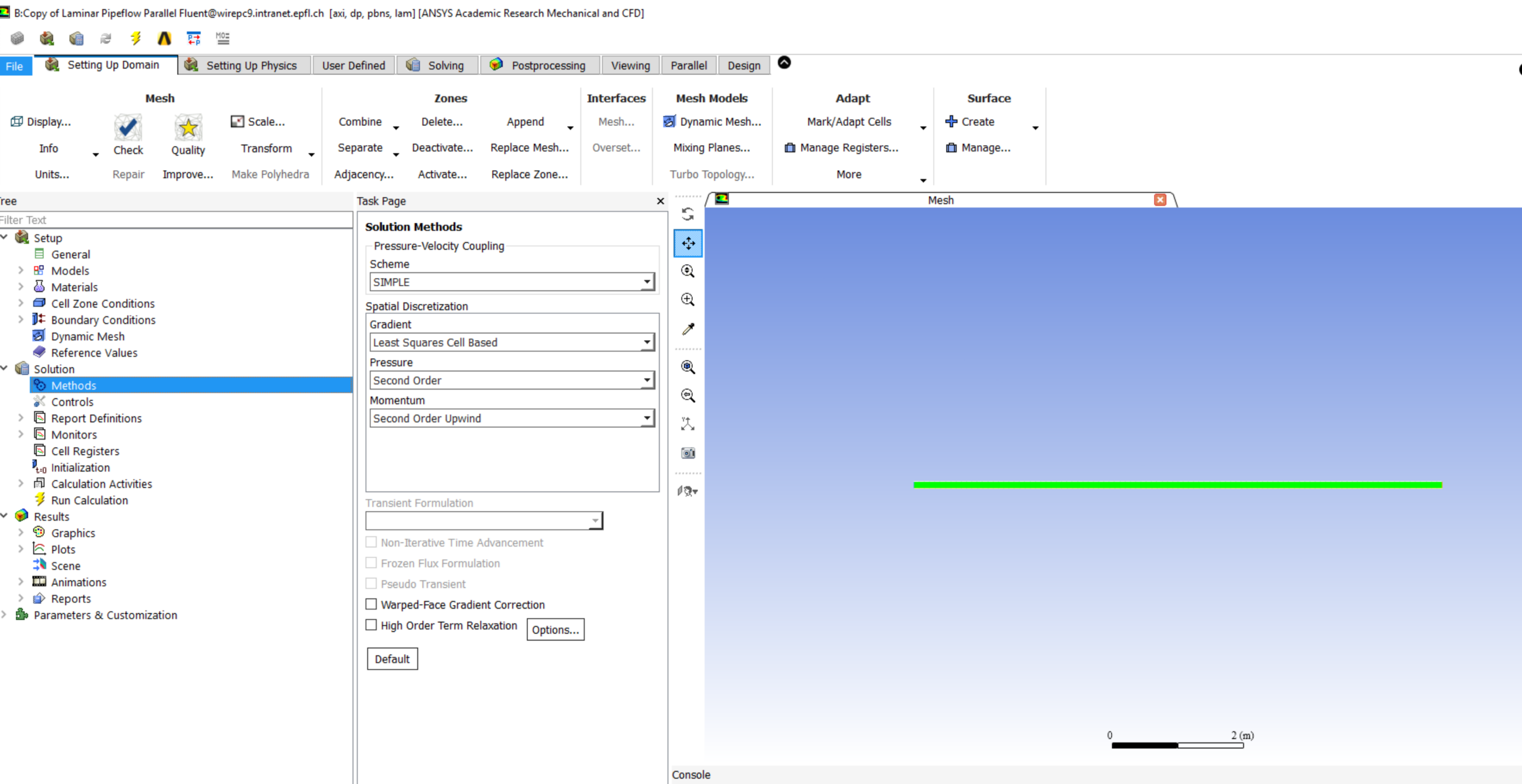


Solution/Methods

At the Tree Menu double click on Solution/**Methods**

Change the Scheme to **SIMPLE**

Check if the Momentum is **Second Order Upwind**

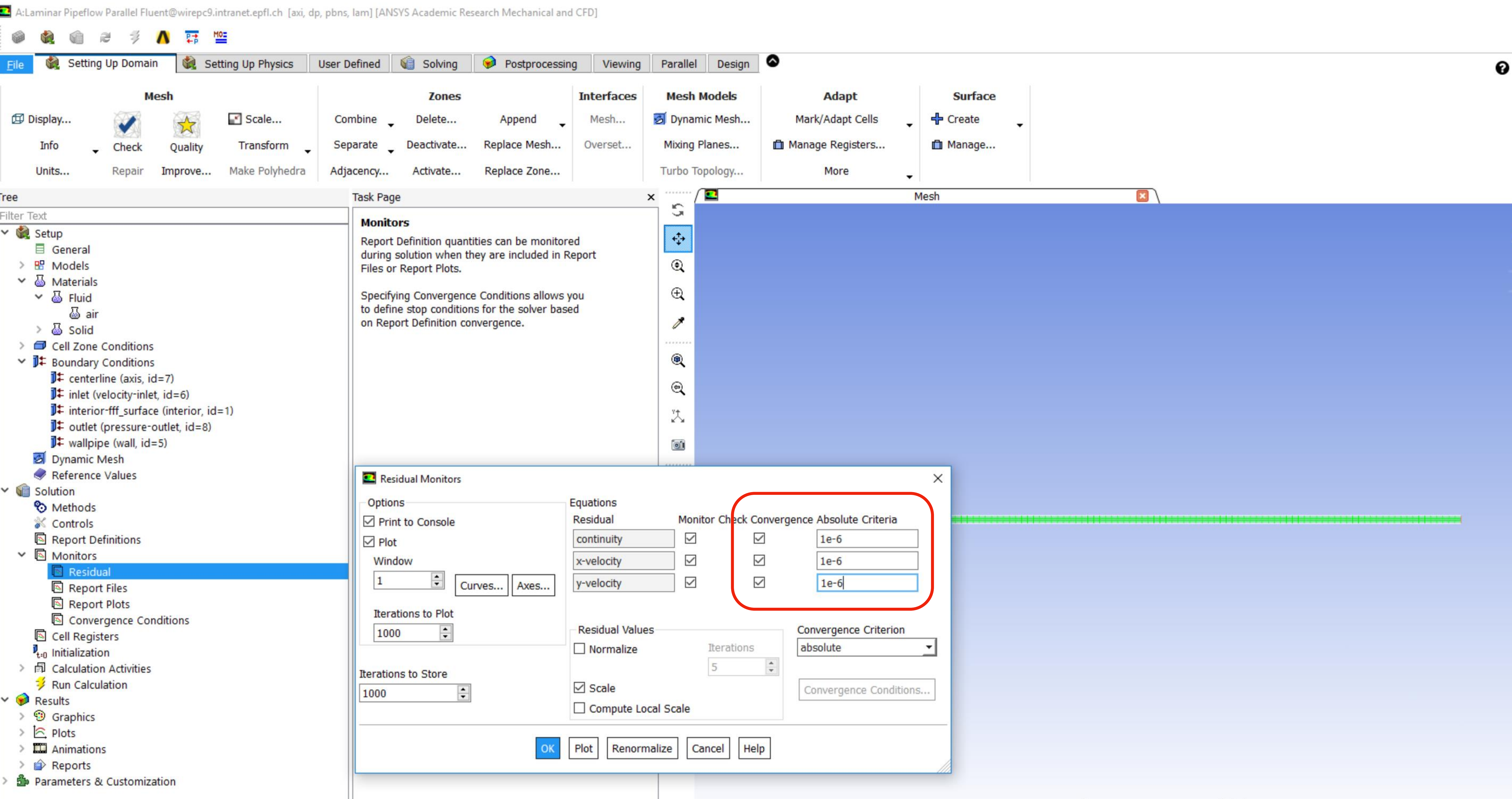


Solution/Monitors (Residuals Control)

At the Tree Menu double click on **Monitors**

Double Click on **Residuals**

Change all the Criteria to **1e-6**



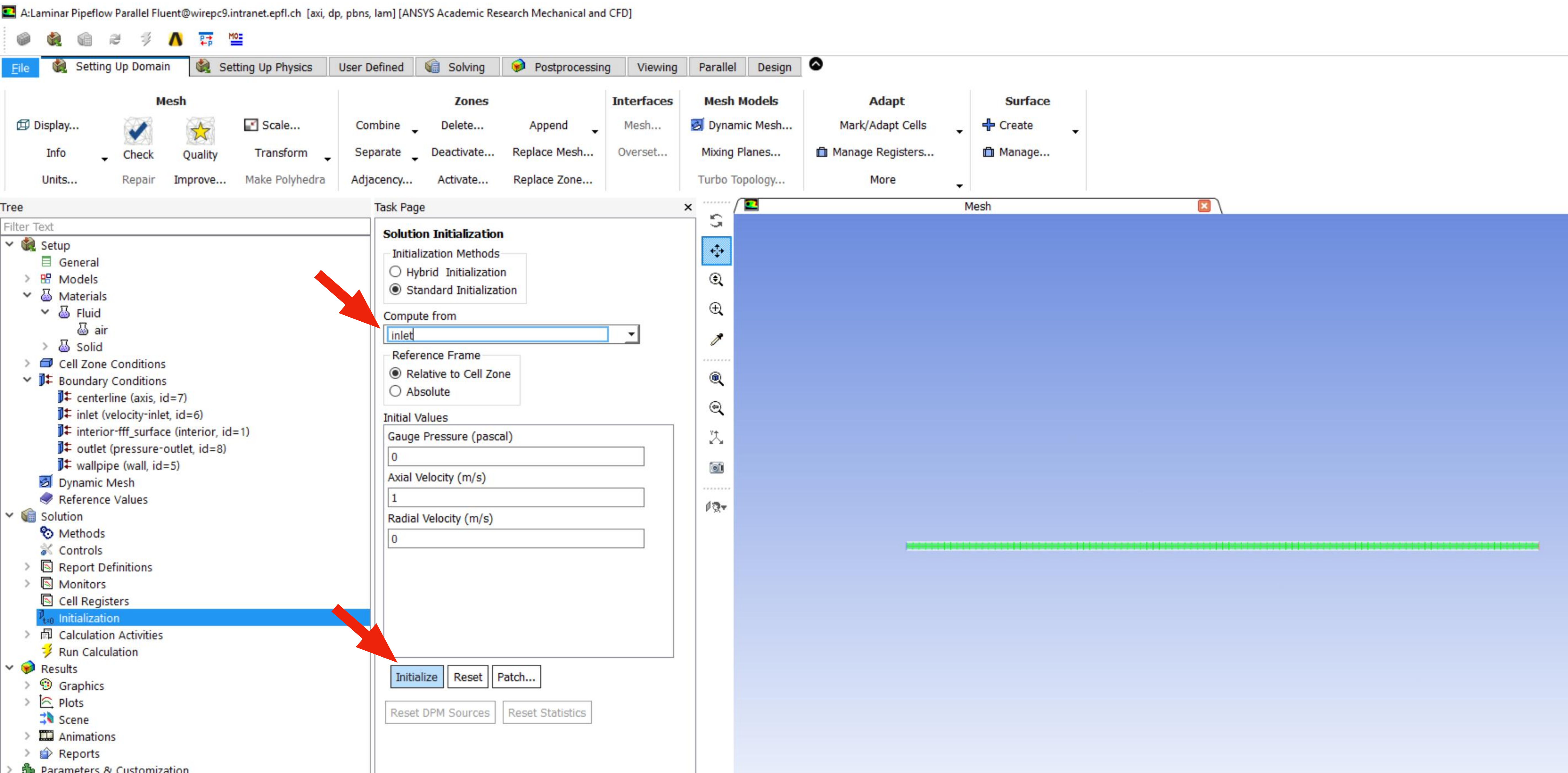
Solution/Initialization

At the Tree Menu double click on **Initialization**

Change the Initialization Methods to **Standard Initialization**

Change the Compute from to **Inlet**

Click on the **Initialize** icon

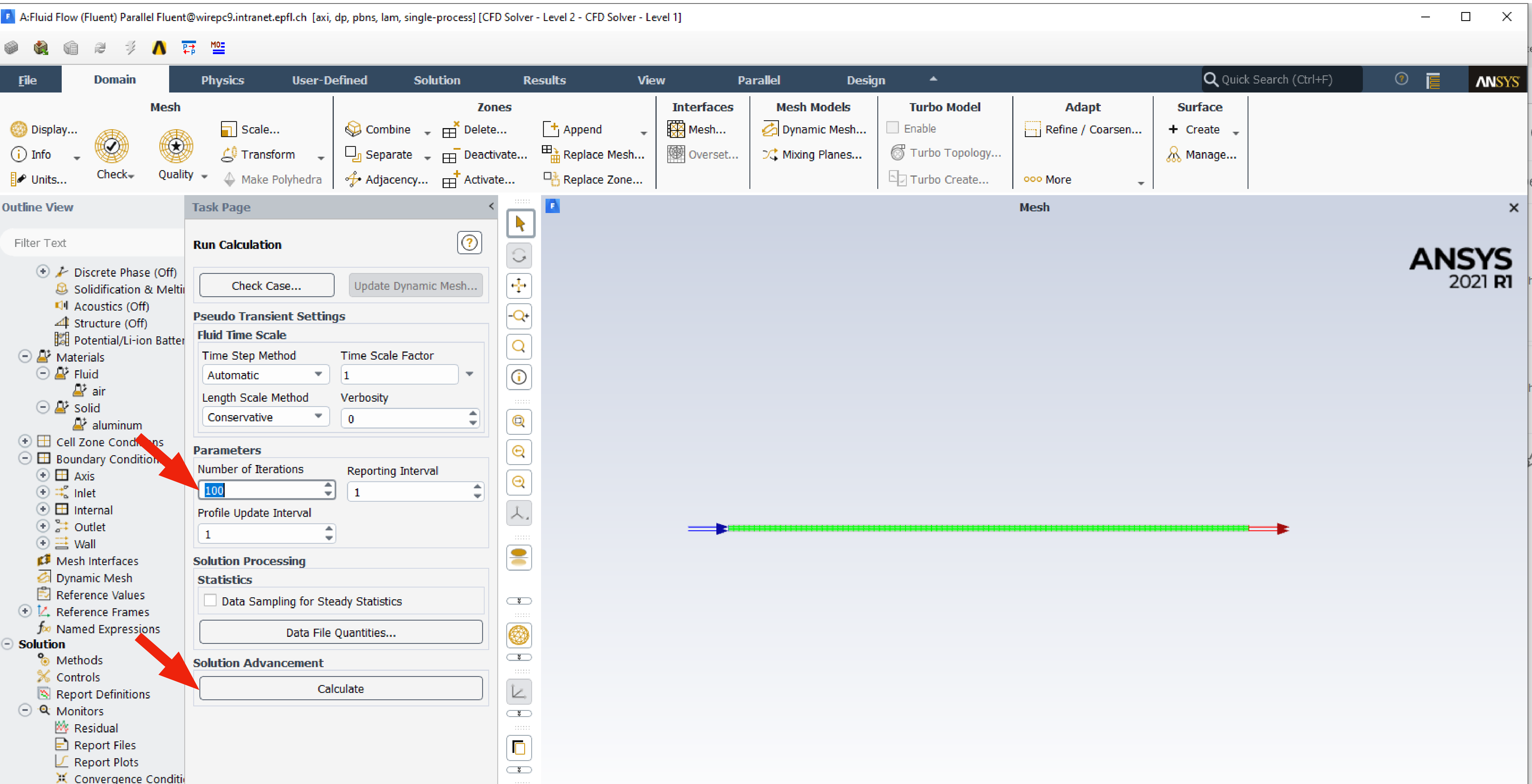


Solution/Run Calculation

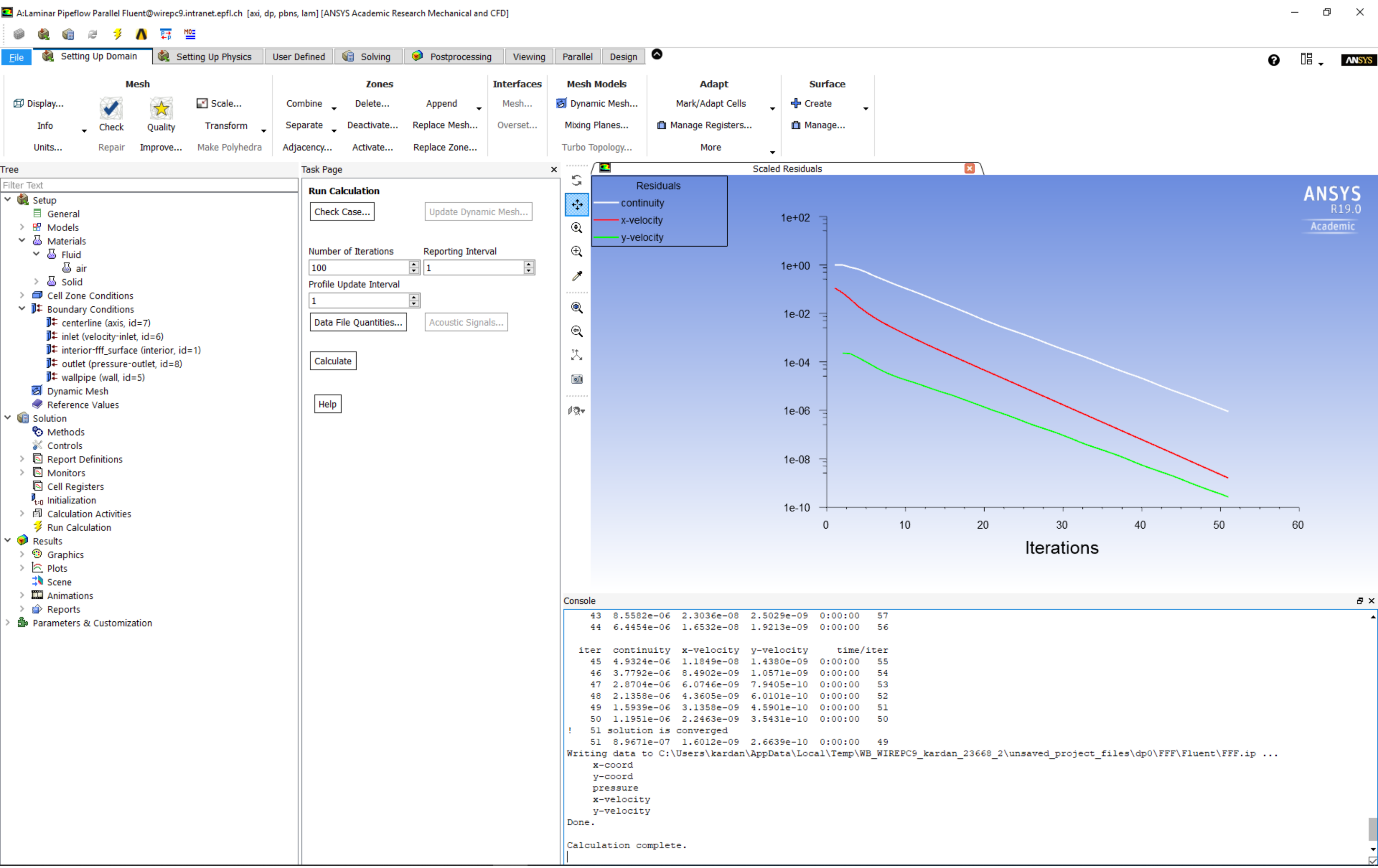
At the Tree Menu double click on **Run Calculation**

Set the Number of Iterations to **100**

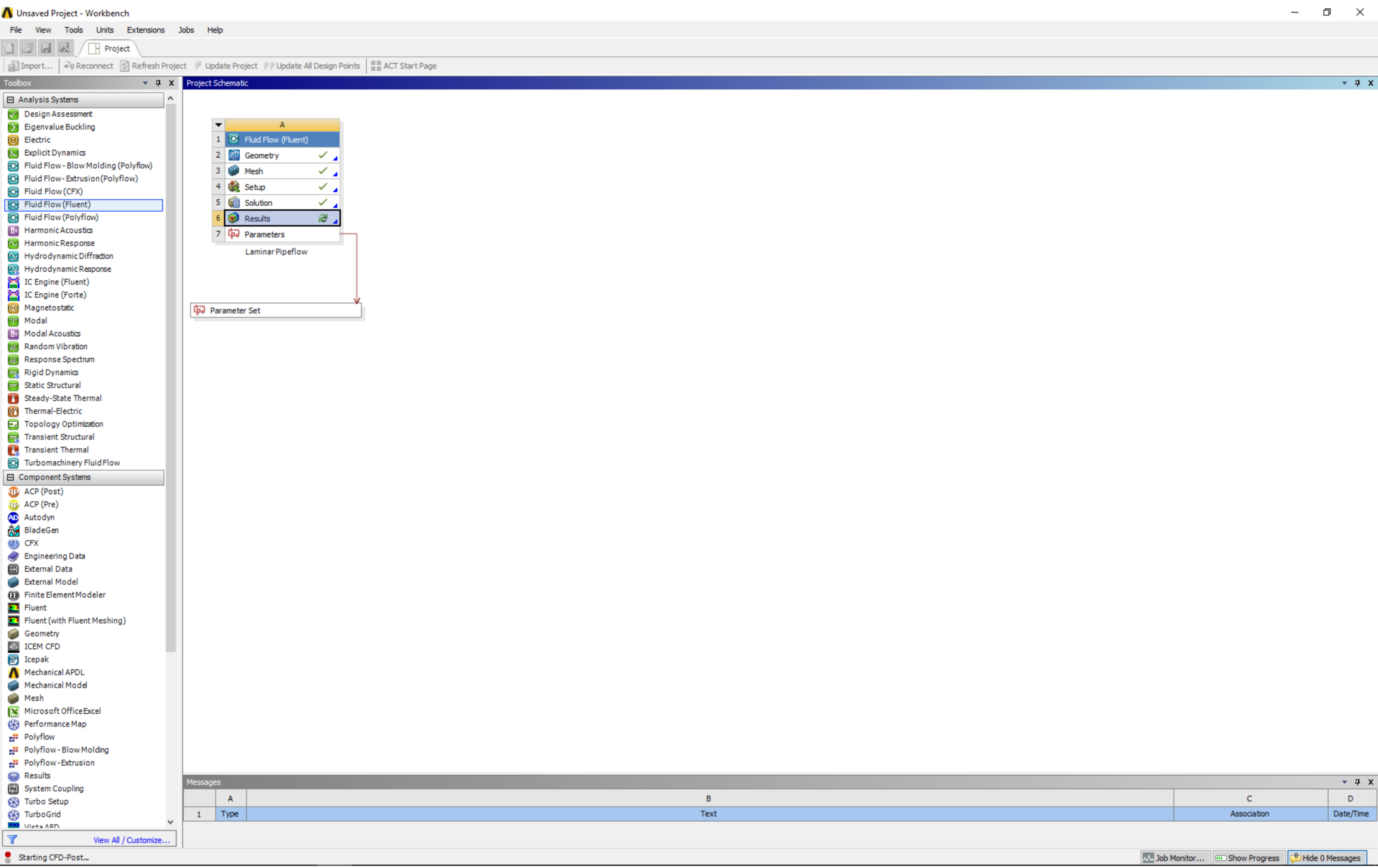
Click on the **Calculate** icon



3- Control residuals

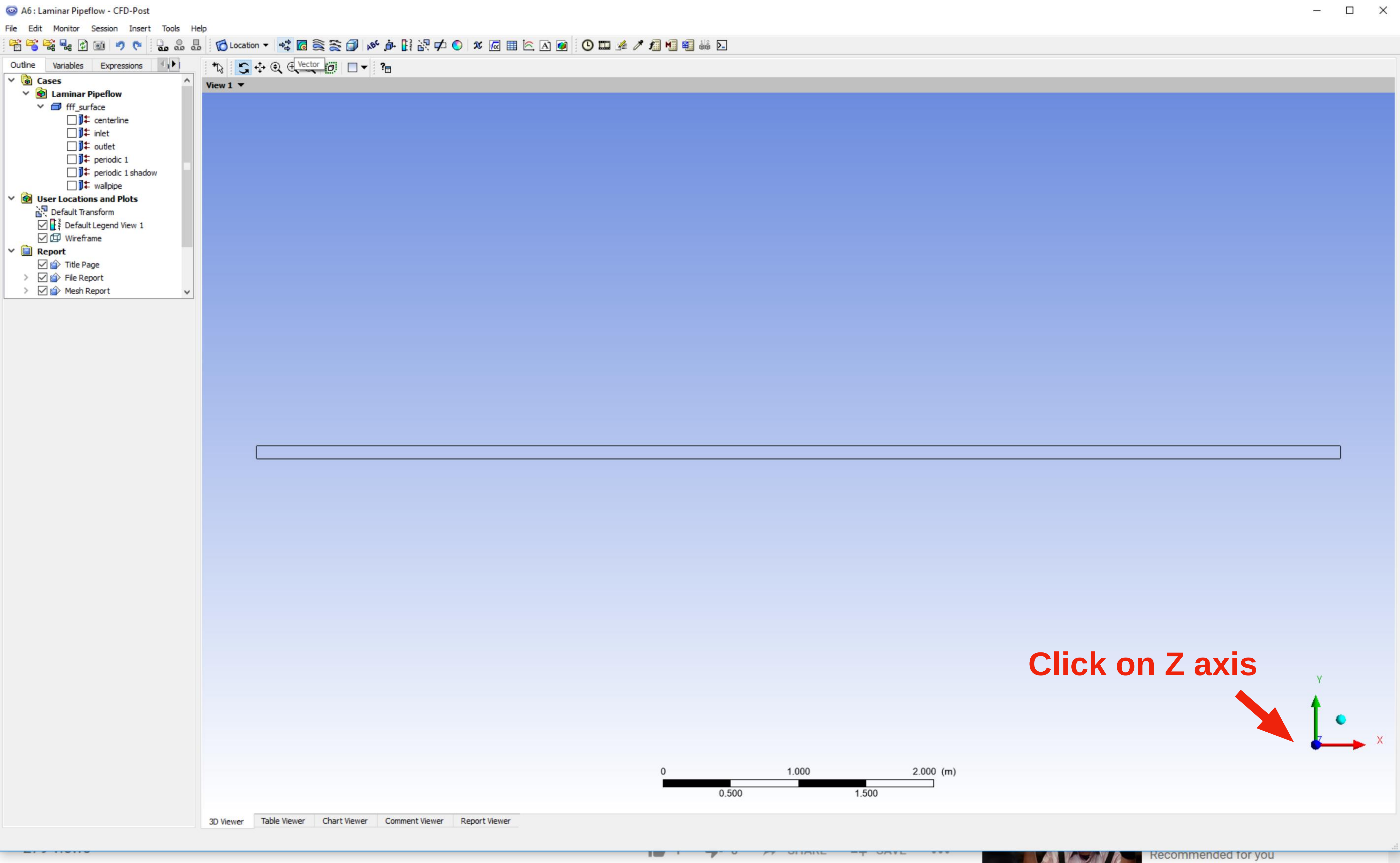


Run Results from Workbench



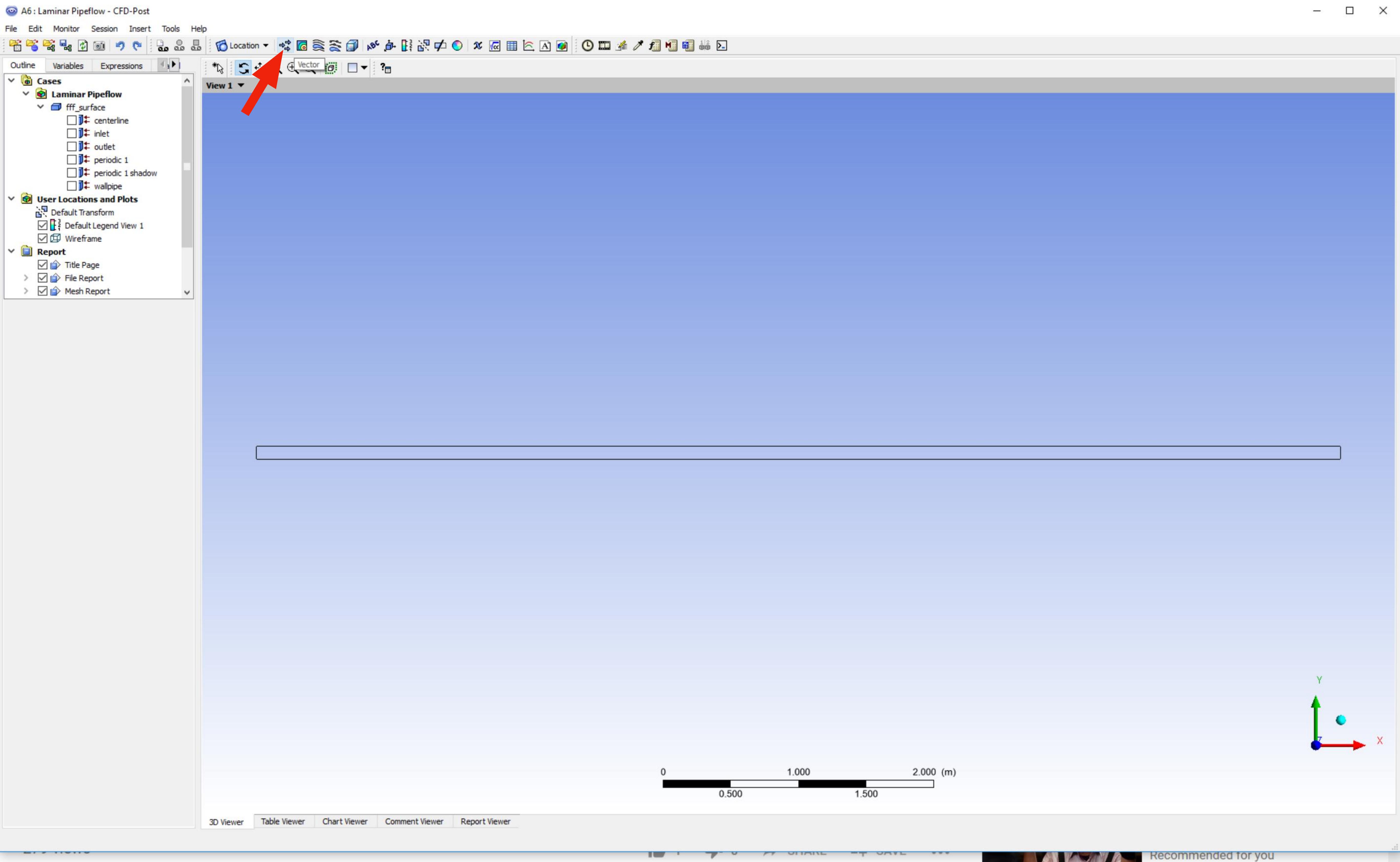
4- CFD-Post (Objective: Velocity Vectors)

Click on Z axis to get x-y view



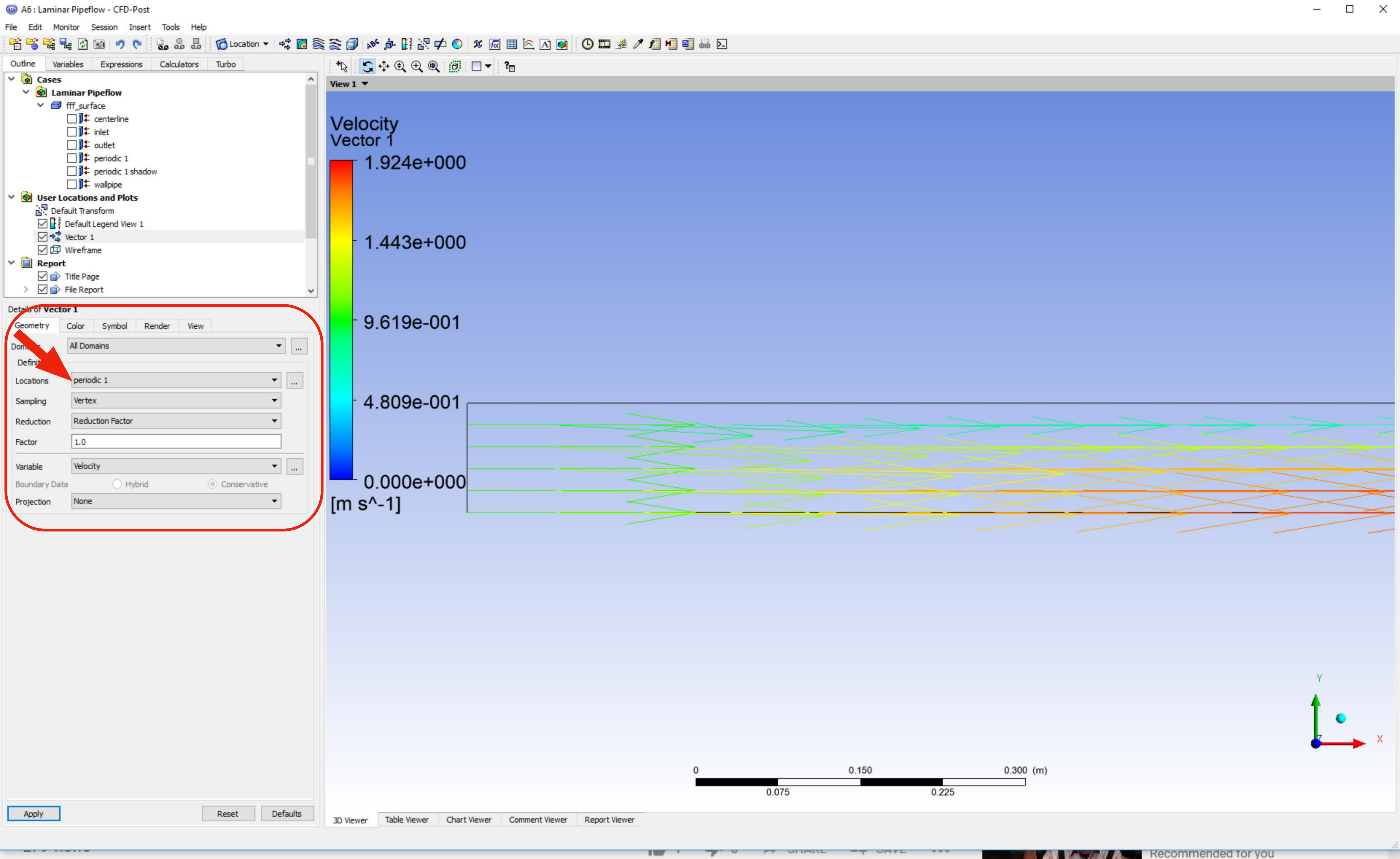
CFD-Post (Objective: Velocity Vectors)

Click on Vector icon then OK



4- CFD-Post (Objective: Velocity Vectors)

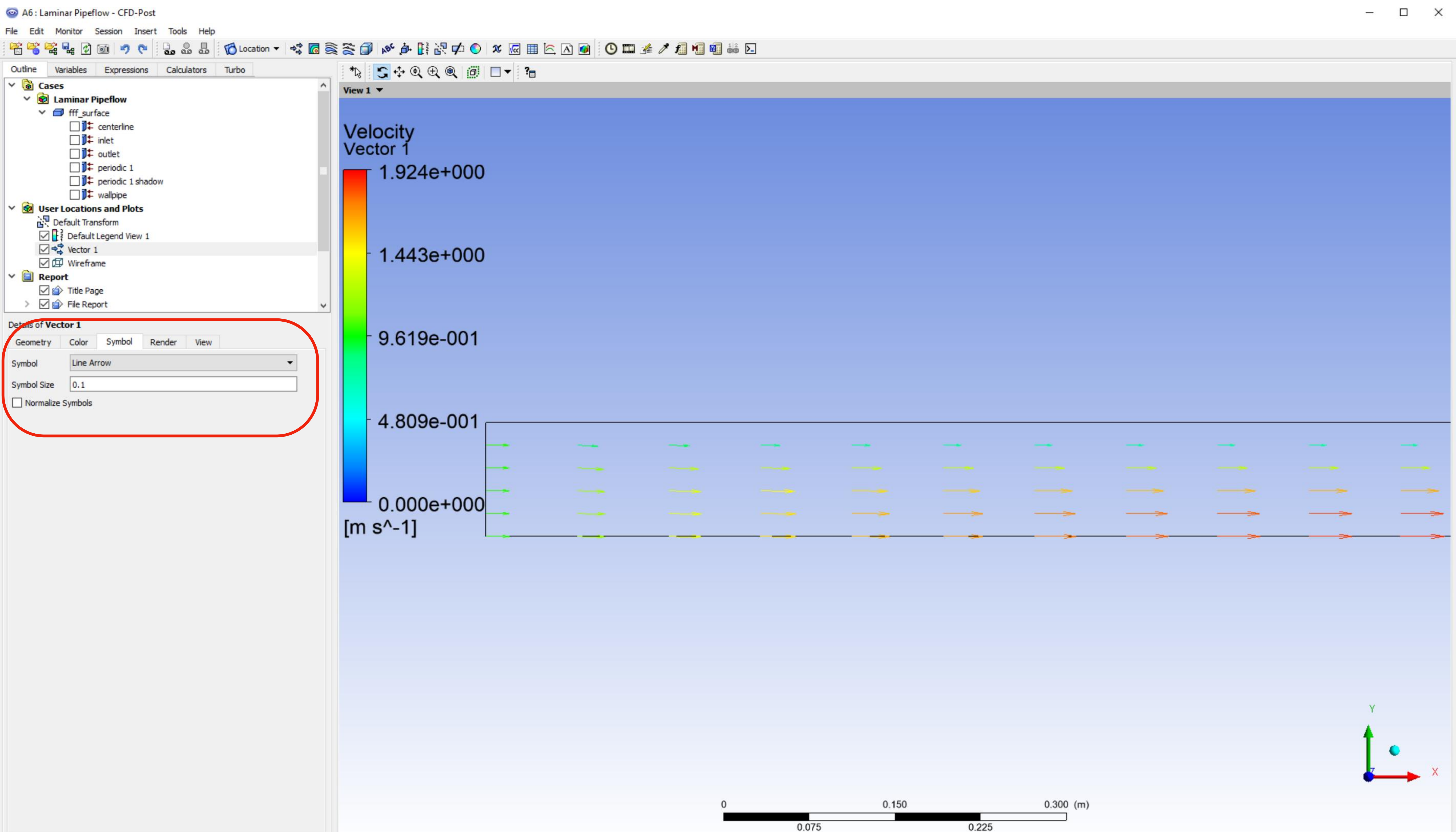
Set the Location to **Periodic 1** then Apply



CFD-Post (Objective: Velocity Vectors)

At the Details Menu click on **Symbol**

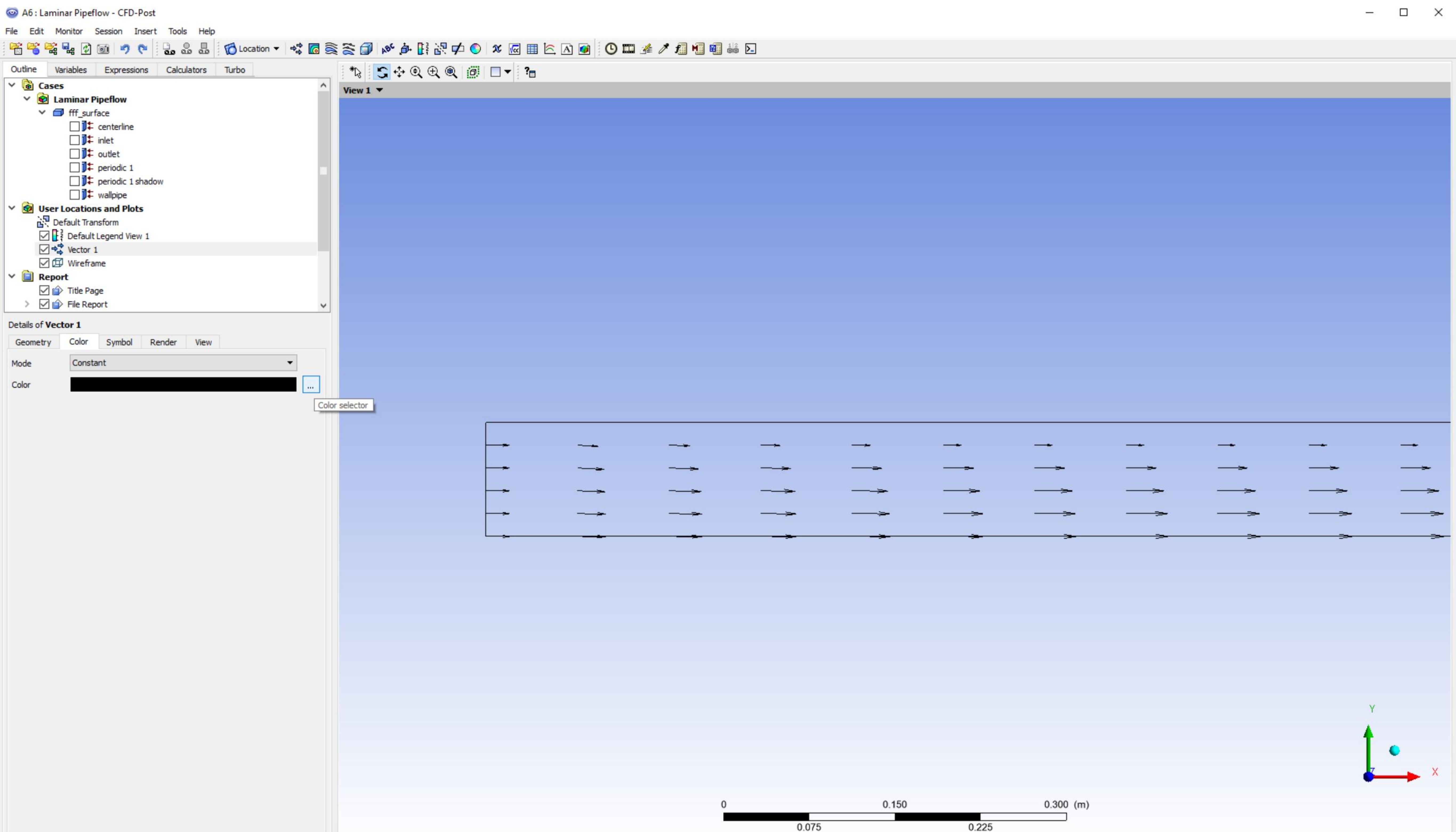
Change the symbol size to **0.1** and Apply



CFD-Post (Objective: Velocity Vectors)

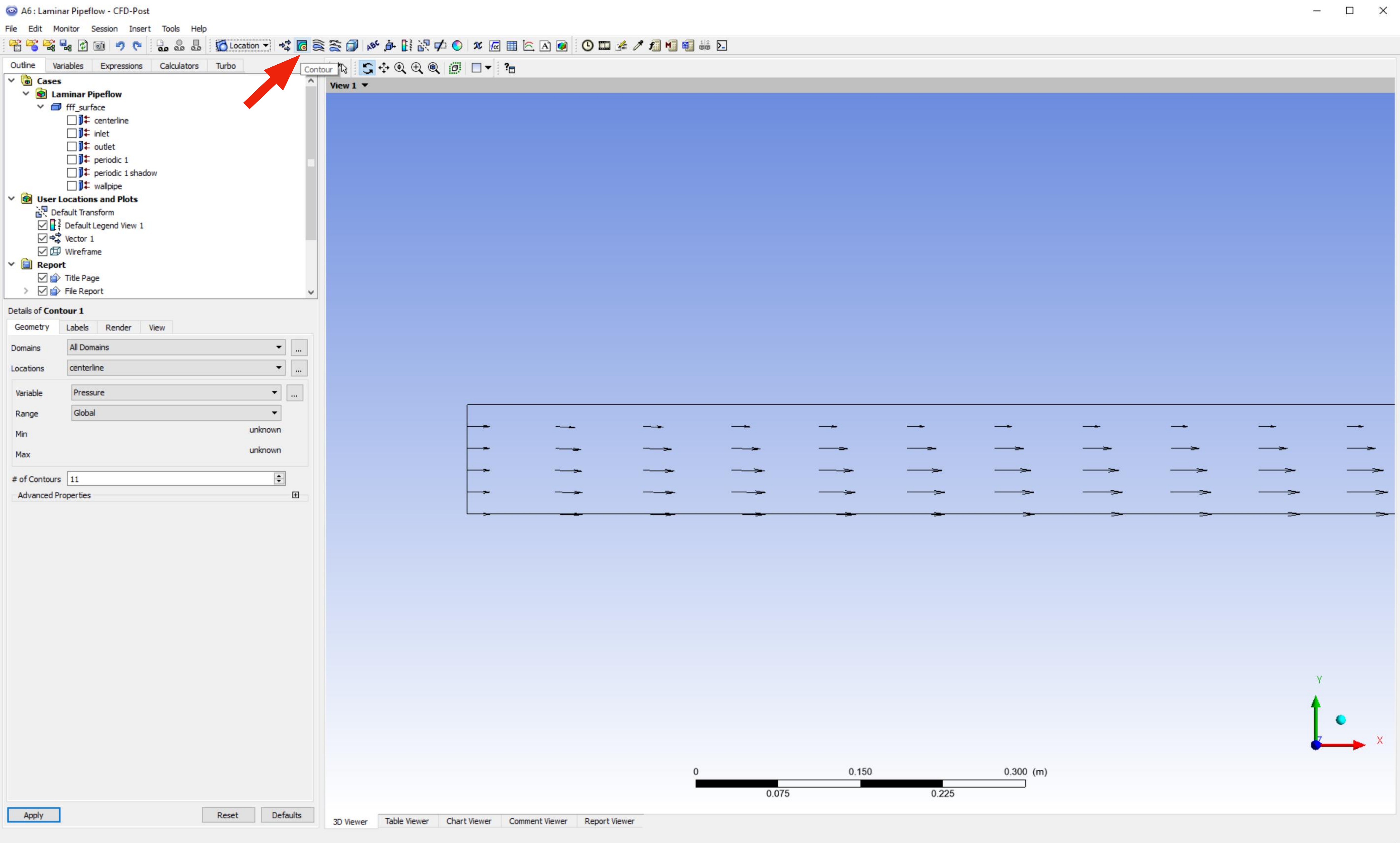
At the Details Menu click on **Color**

Change the Mode to **Constant** and define black color, then Apply



CFD-Post (Objective: Velocity Contour)

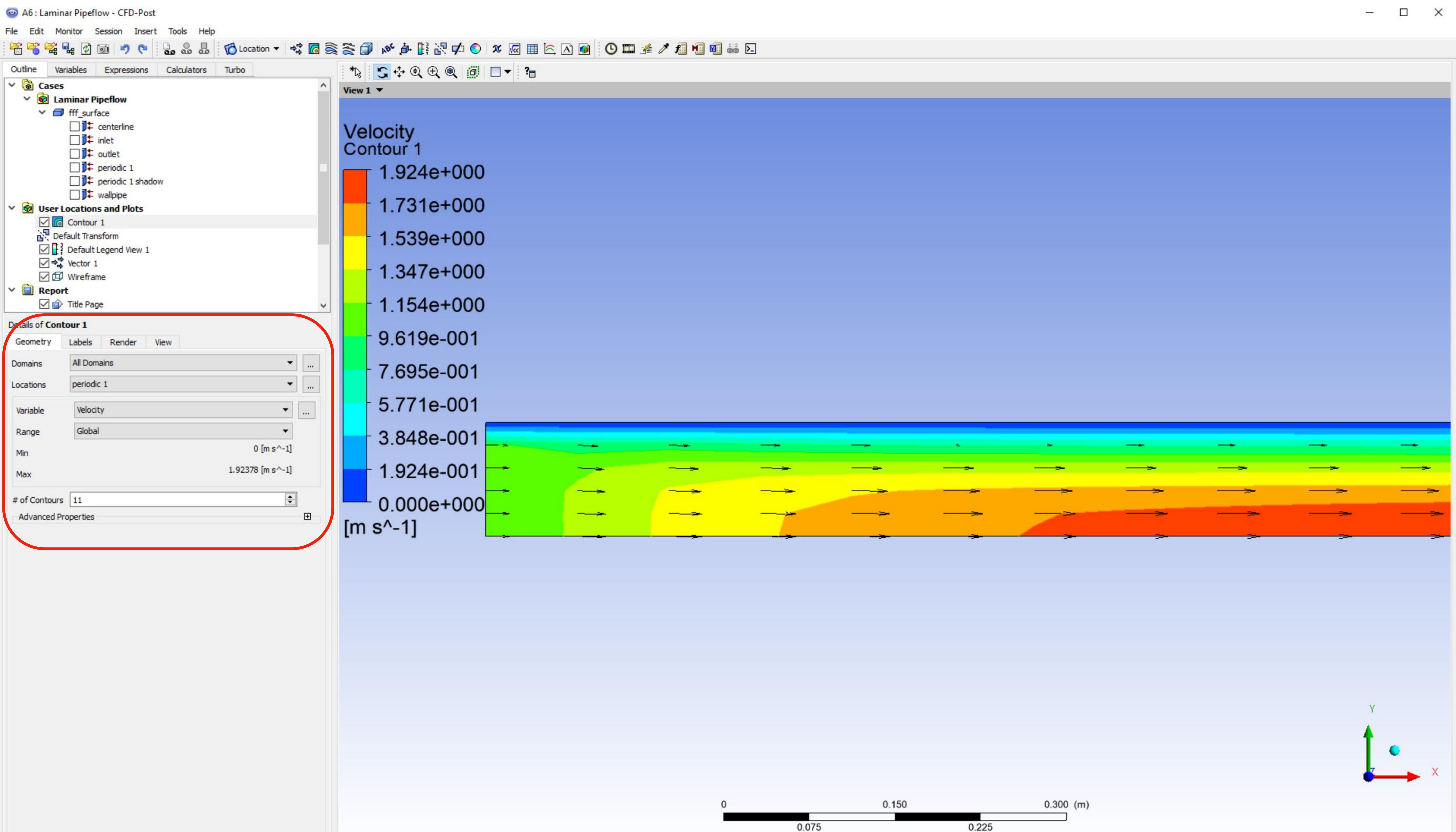
Click on Contour icon then OK



CFD-Post (Objective: Velocity Contour)

Set the Location to **Periodic 1**

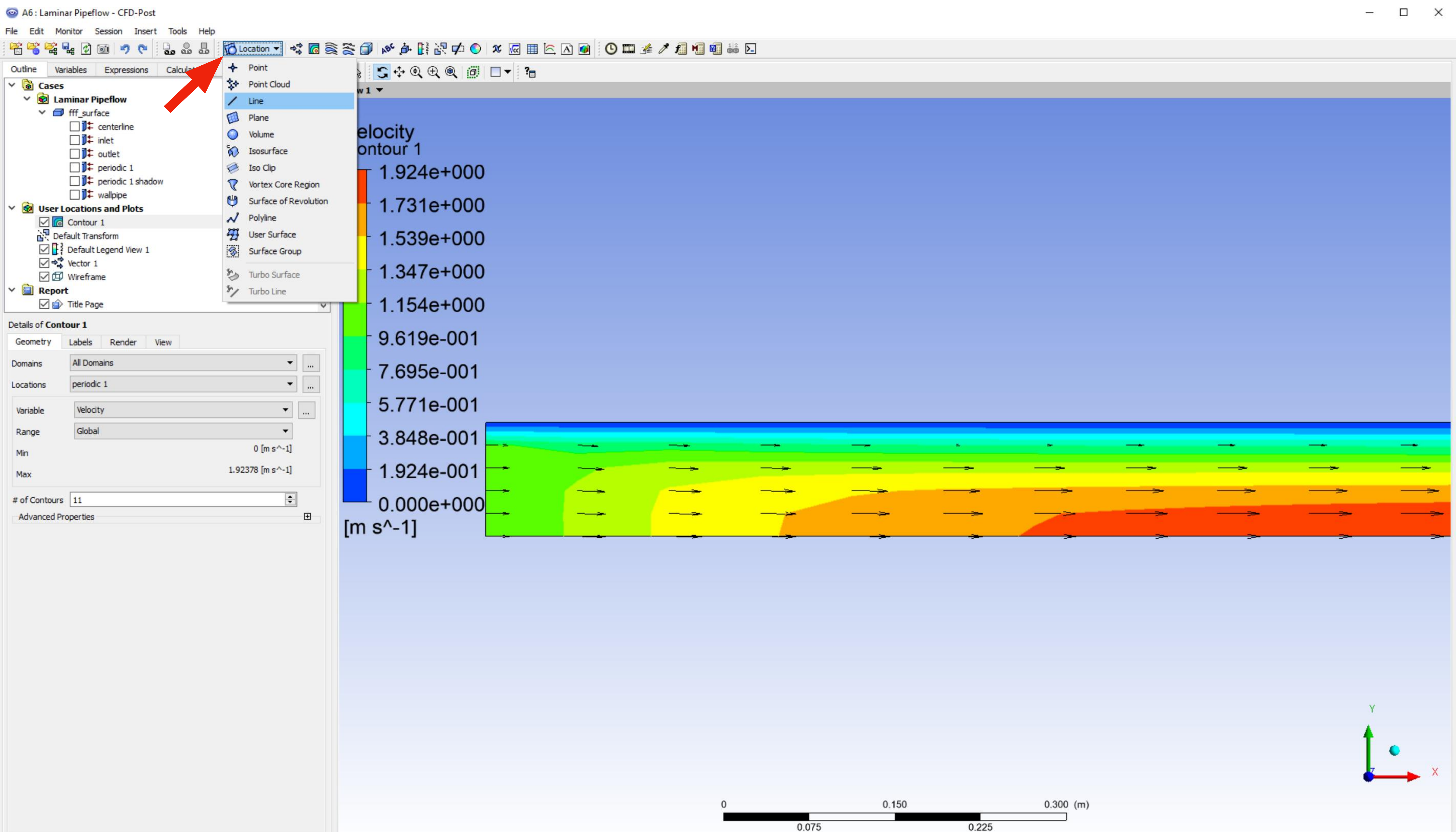
Change the Variable to **Velocity**, then Apply



CFD-Post (Objective: Creating Chart)

Location/Line

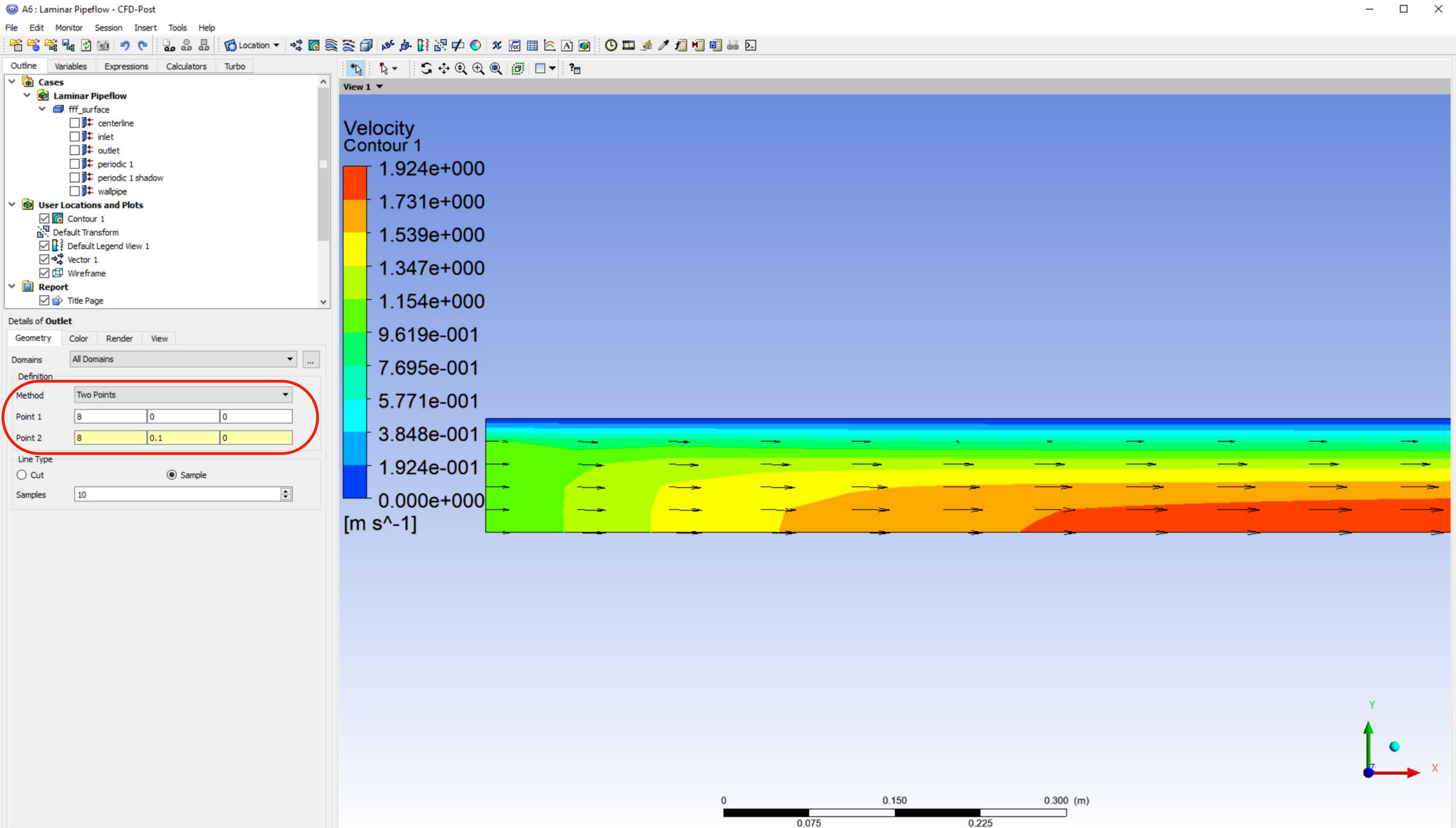
Change the Name to **Outlet**, then Ok



CFD-Post (Objective: Creating Chart)

Set the Point 1 to **(8)(0)(0)**

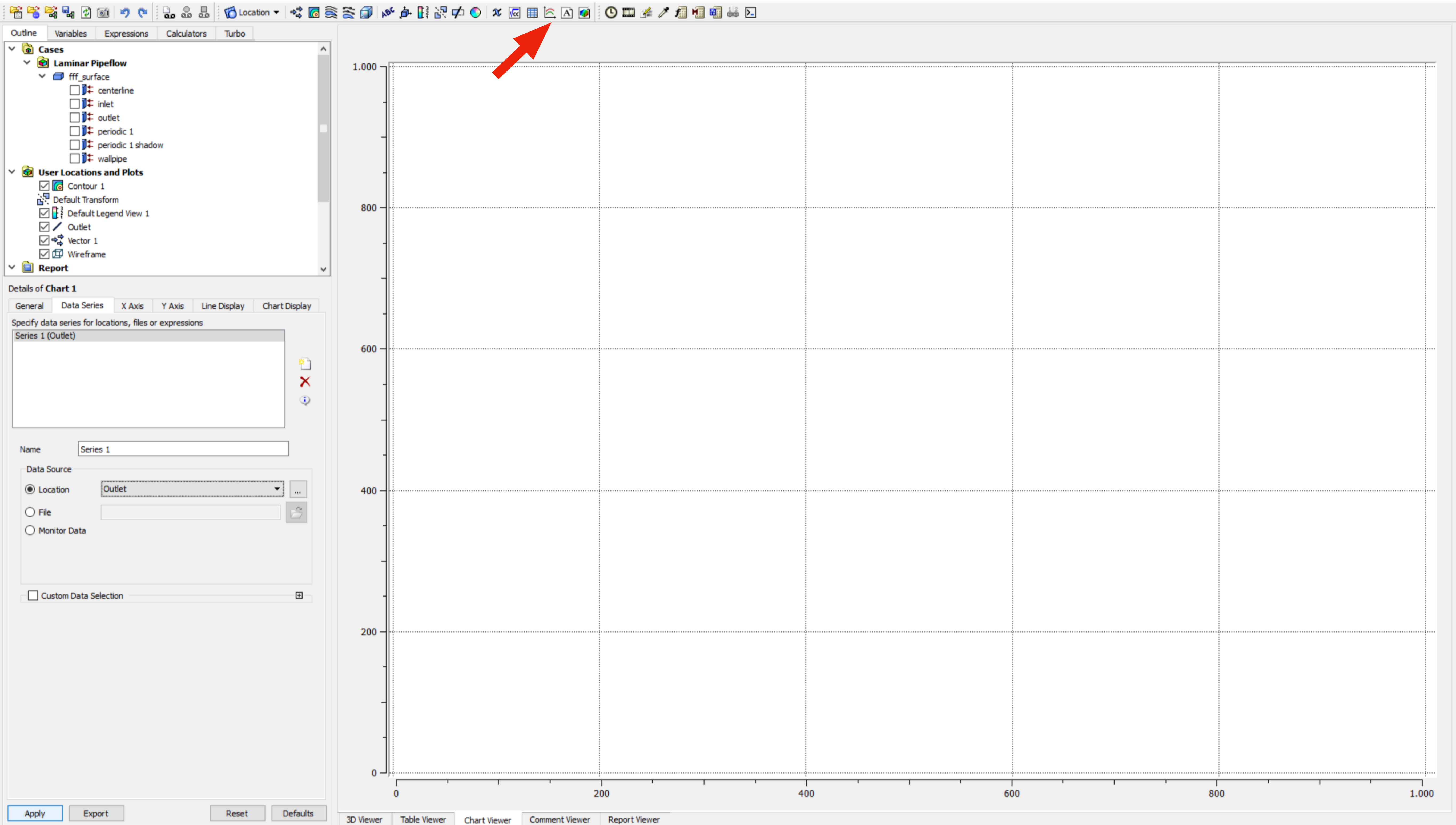
Set the Point 2 to **(8)(0.1)(0)**, then Apply



CFD-Post (Objective: Creating Chart)

Click on Chart

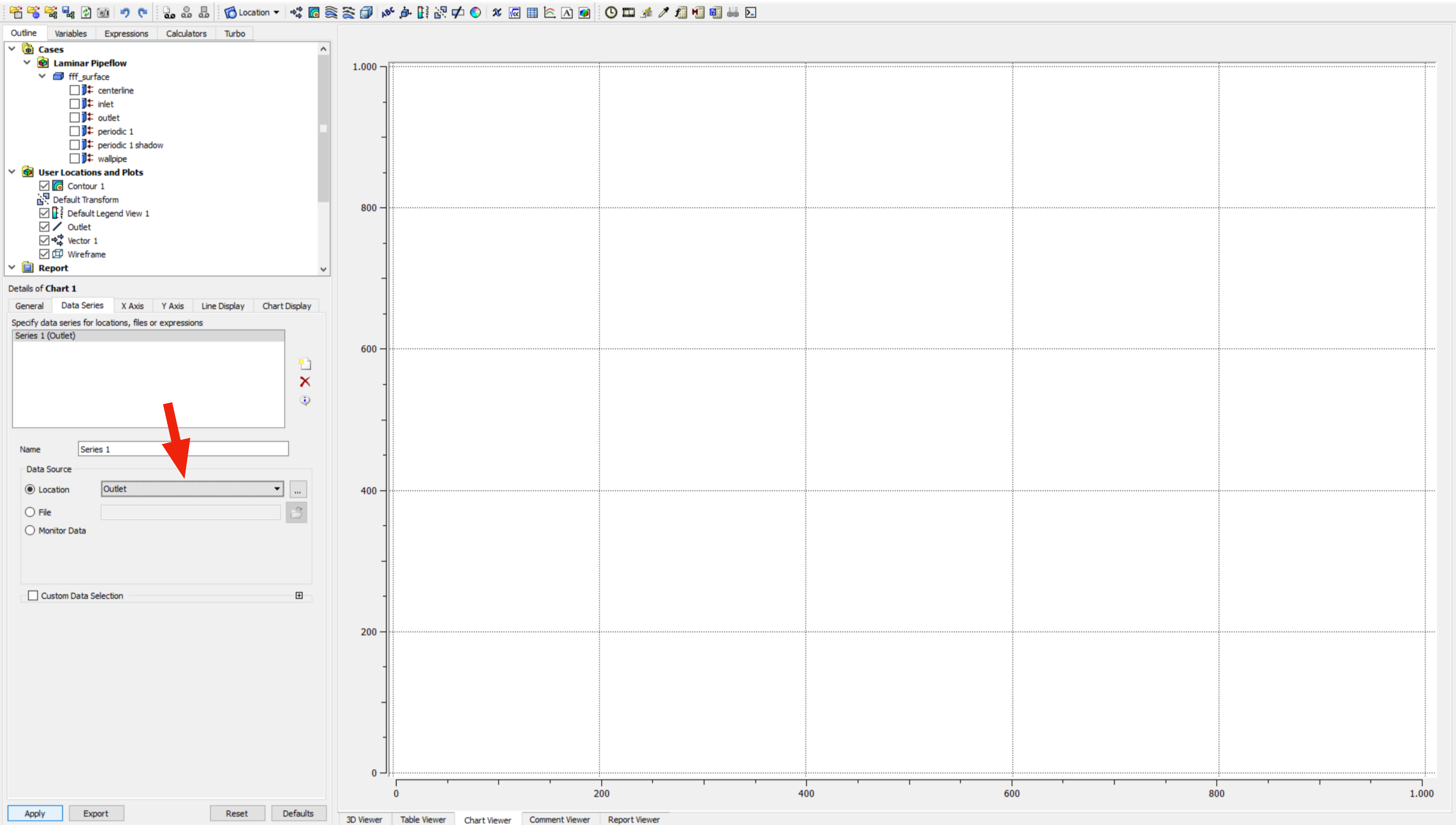
Change the Name to **Velocity Out**, then Ok



CFD-Post (Objective: Creating Chart)

At the Details Menu click on **Data Series**

Change the Location to **Outlet**, then Apply



CFD-Post (Objective: Creating Chart)

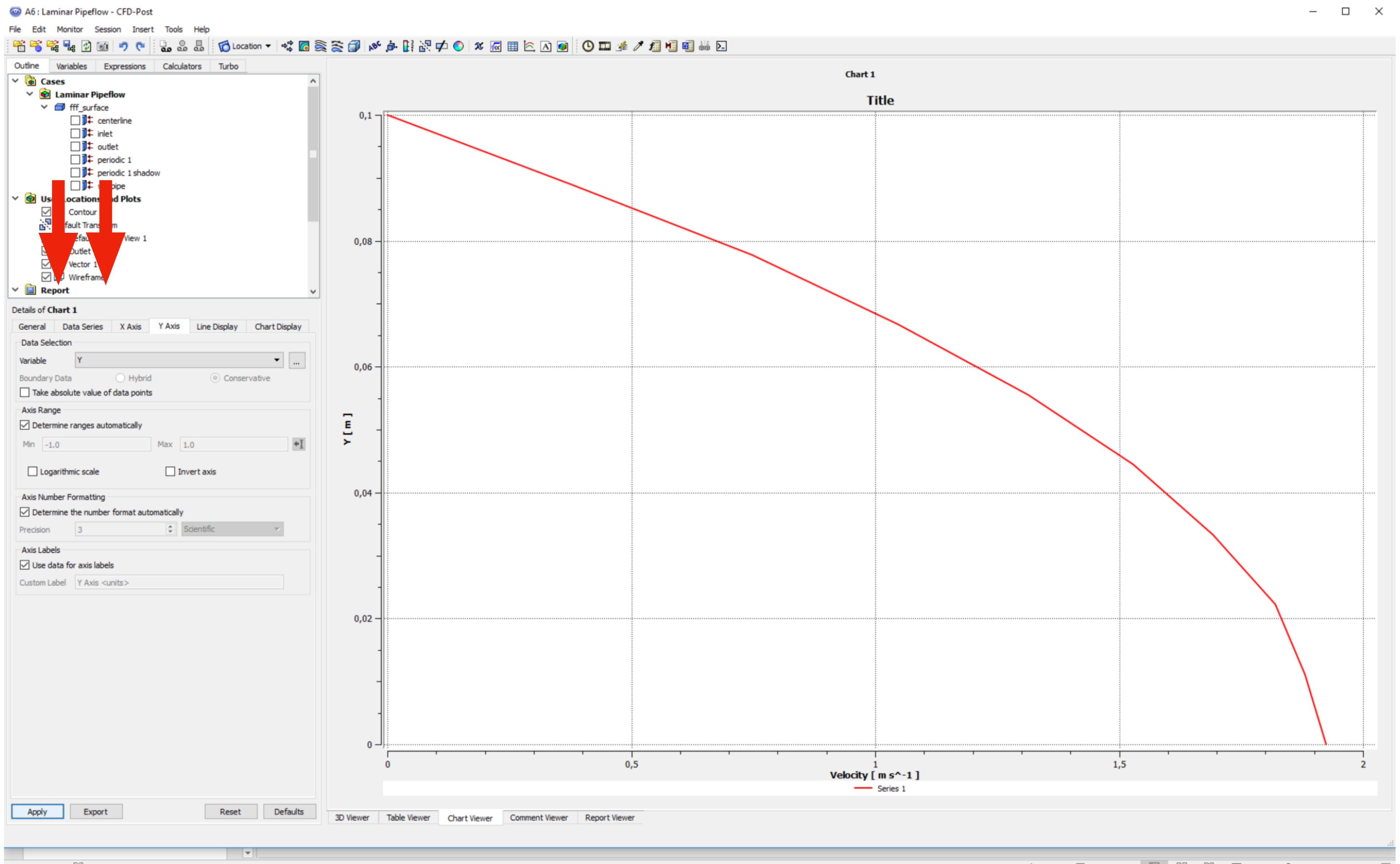
At the Details Menu click on **X Axis**

Change the Variable to **Velocity**

At the Details Menu click on **Y Axis**

Change the Variable to **Y**, then Apply

Close the CFD-Post software

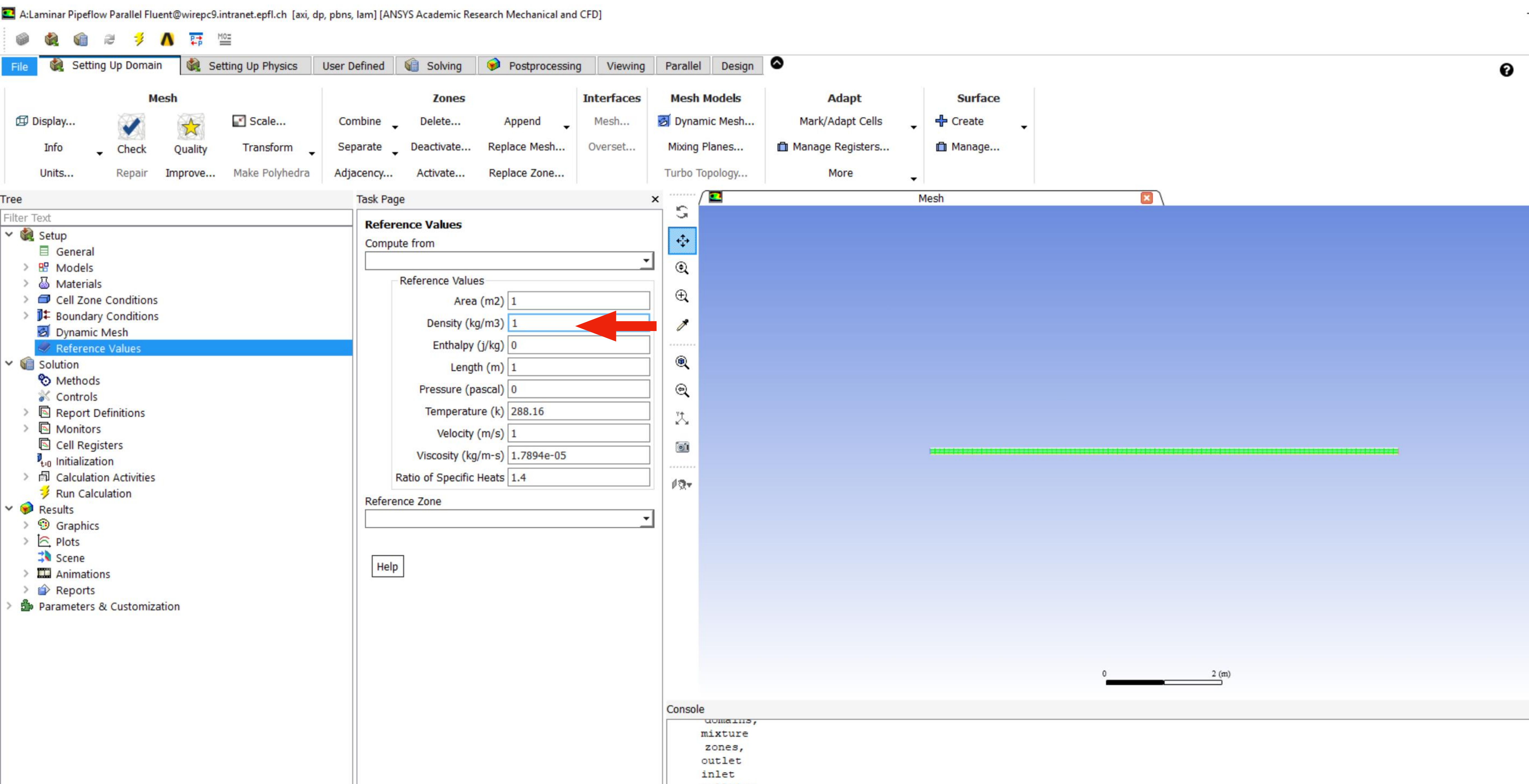


CFD-Post (Objective: Wall Shear)

Double click on **Solution** from **Workbench** (Run Fluent)

At the Tree Menu double click on **Reference Values**

Set Velocity and Density to **1**

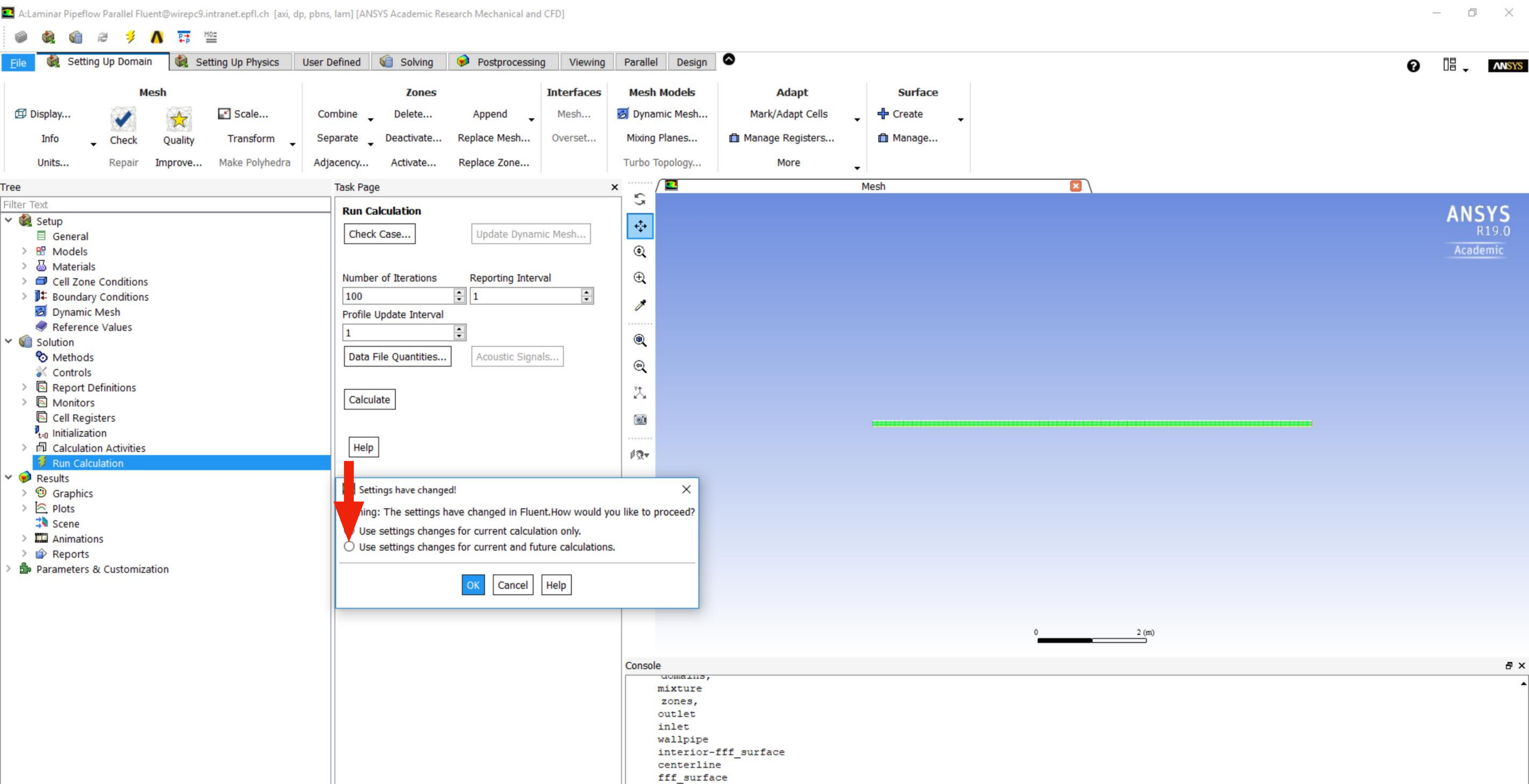


CFD-Post (Objective: Wall Shear)

At the Tree Menu double click on **Run Calculation**

Click on the **Calculate** icon

Select **Use setting changes for current and future calculations**, then OK



CFD-Post (Objective: Wall Shear)

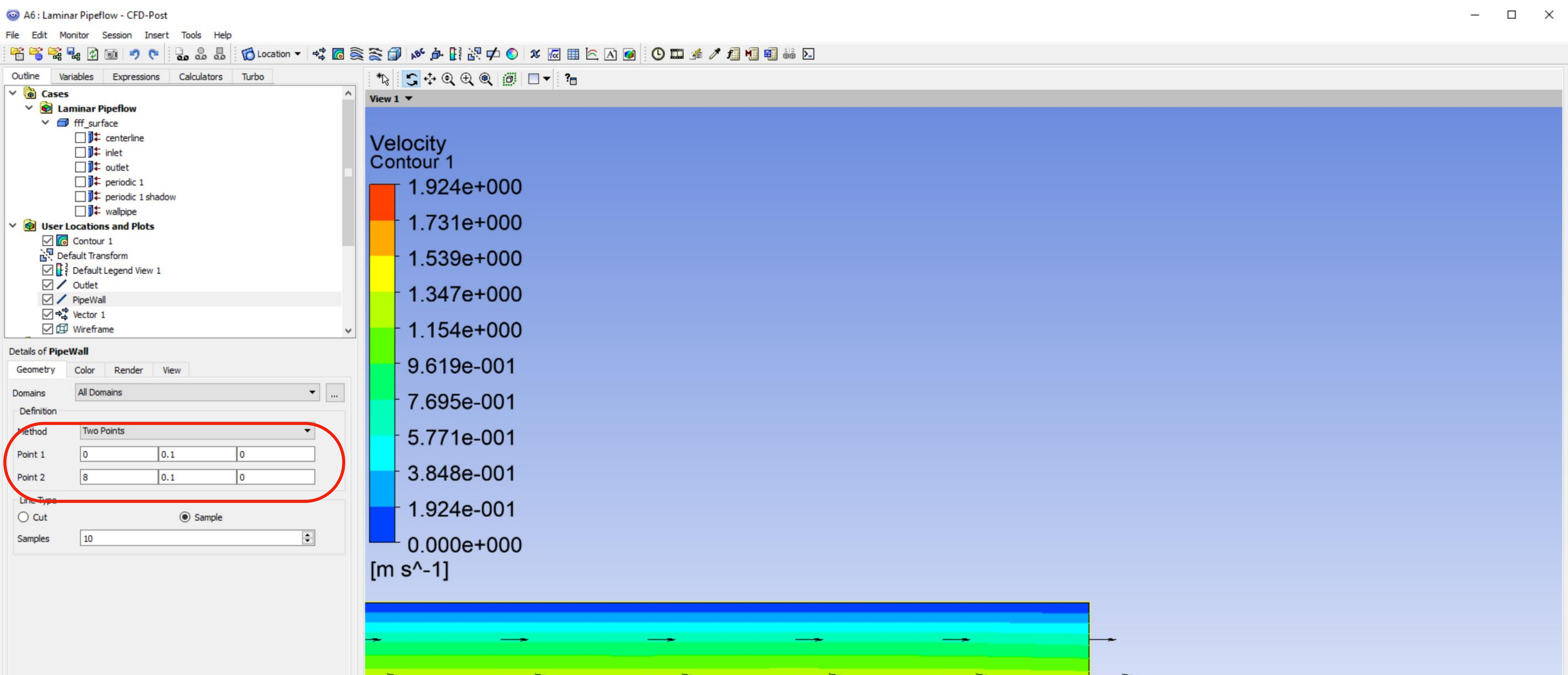
Double click on Results from Workbench

Location/Line

Change the Name to **PipeWall**, then Ok

Set the Point 1 to **(0)(0.1)(0)**

Set the Point 2 to **(8)(0.1)(0)**, then Apply



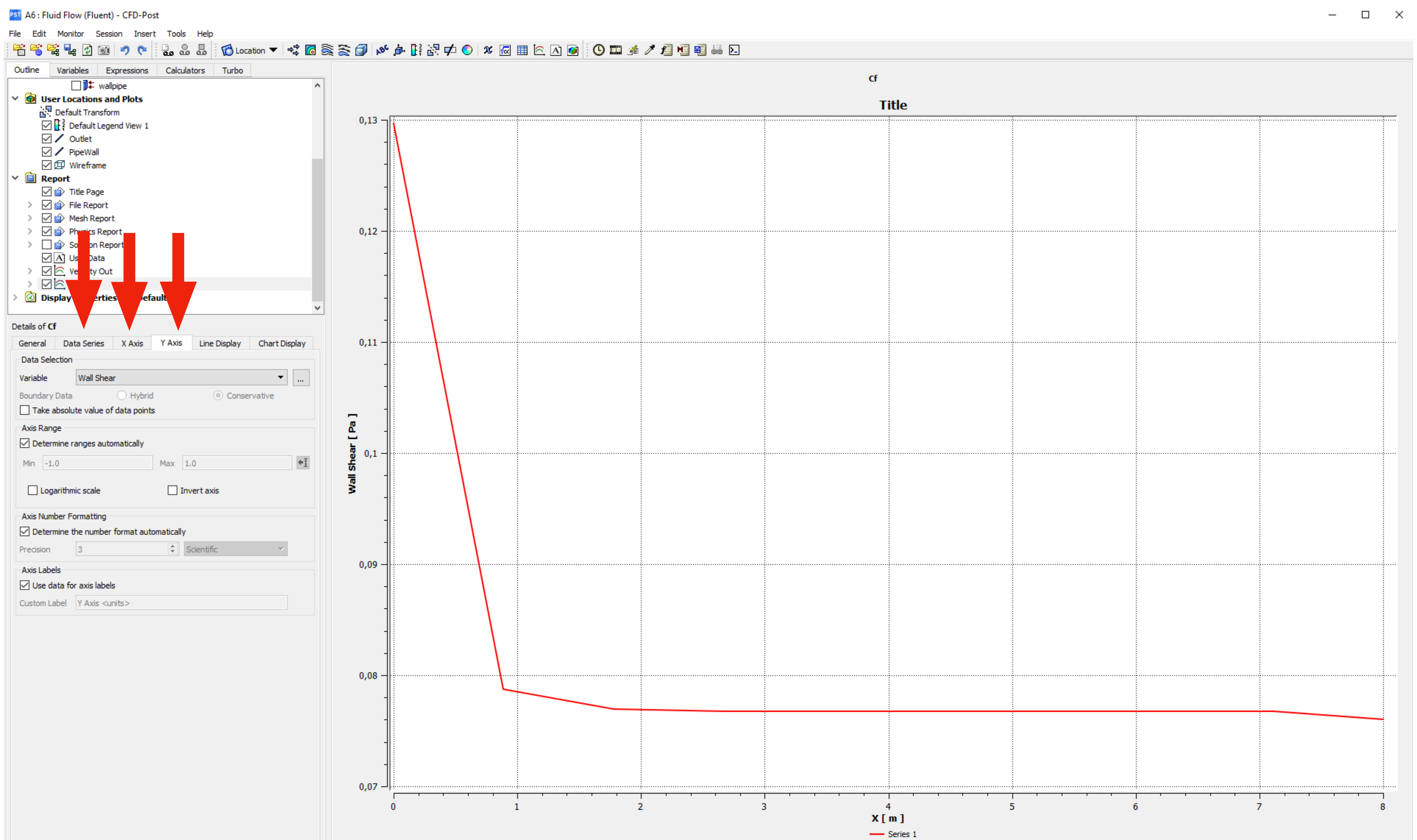
CFD-Post (Objective: Wall Shear)

Click on Chart/Change the Name to **Wall Shear**, then OK

At the Details Menu, Click on **Data Series**, Change the Location to **PipeWall**

Click on **X Axis,** then Change the Variable to **X**

Click on **Y Axis,** Change the Variable to **Wall Shear**, Apply



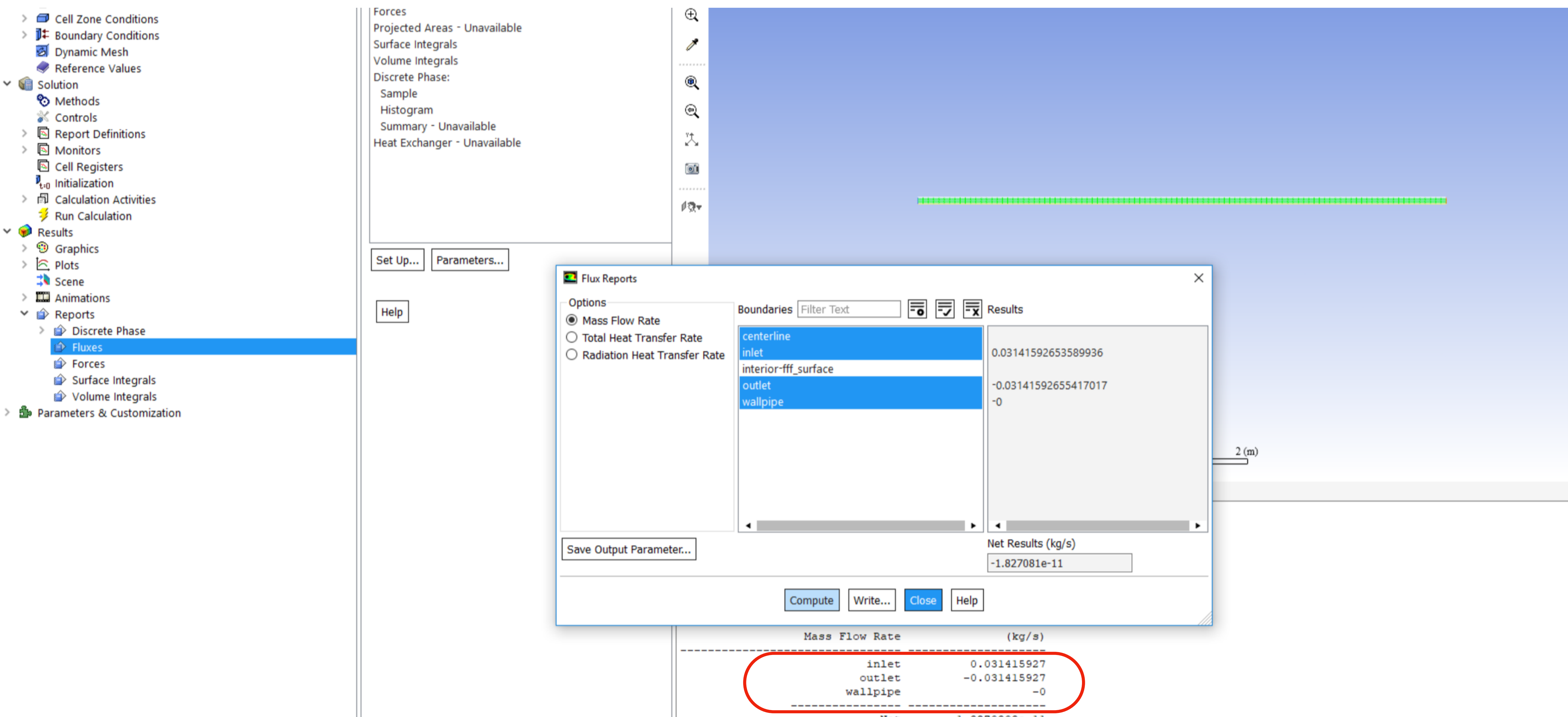
Fluent (Results Reports)

Double click on **Solution** from Workbench to open Fluent

At the Tree Menu select Results/Reports/**Fluxes** (D-click on Fluxes)

Select **centerline, inlet, outlet and wallpipe**, then Compute

Mass Flow Rate will be calculated and appeared at Console box

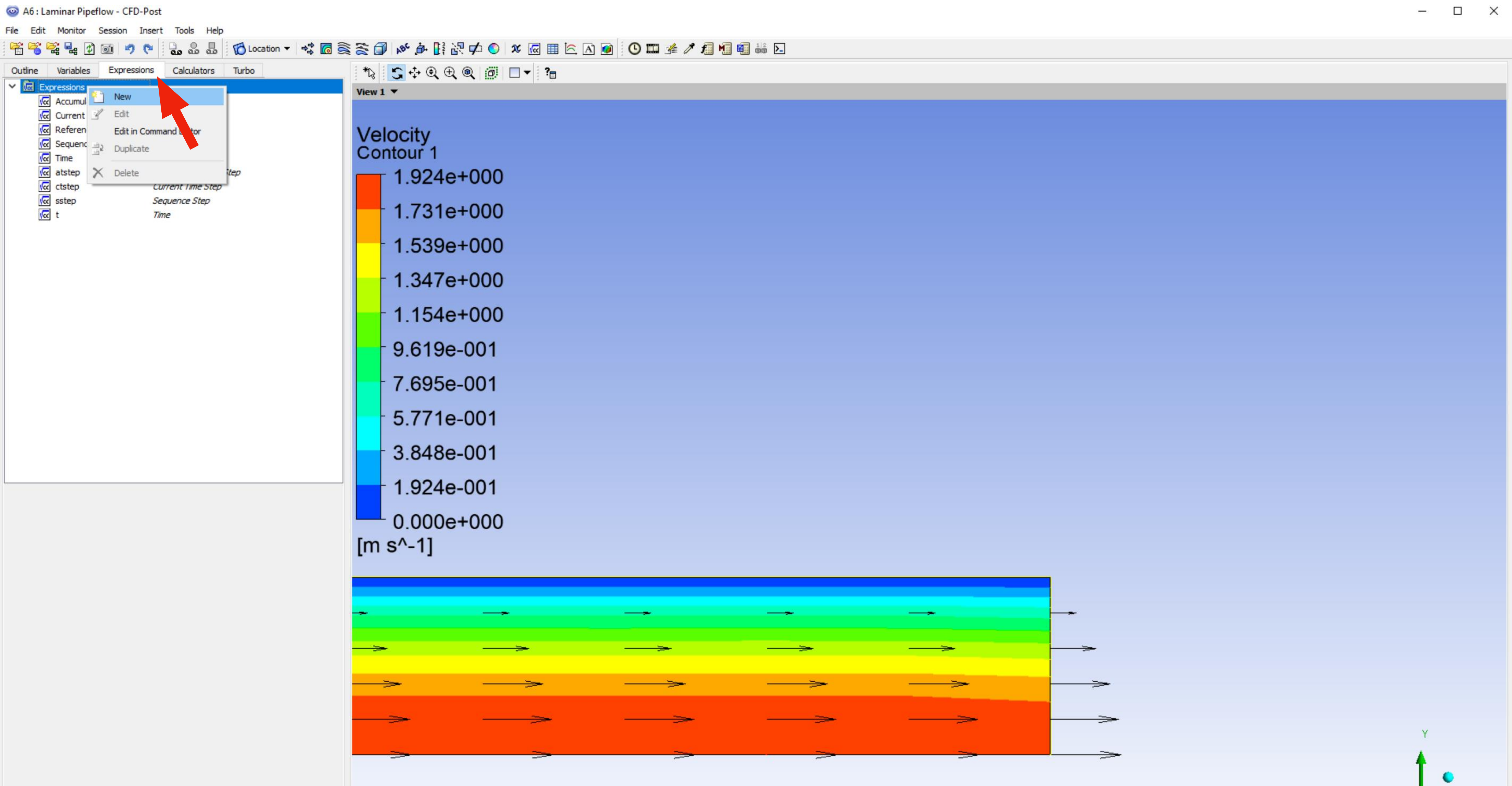


CFD-Post (Objective: Expressions, Momentum Difference)

Click on **Expressions** tab, then Ok

R-Click on Expressions, then click on New

Change the Name to **LHS**, then Ok



CFD-Post (Objective: Expressions, Momentum Balance)

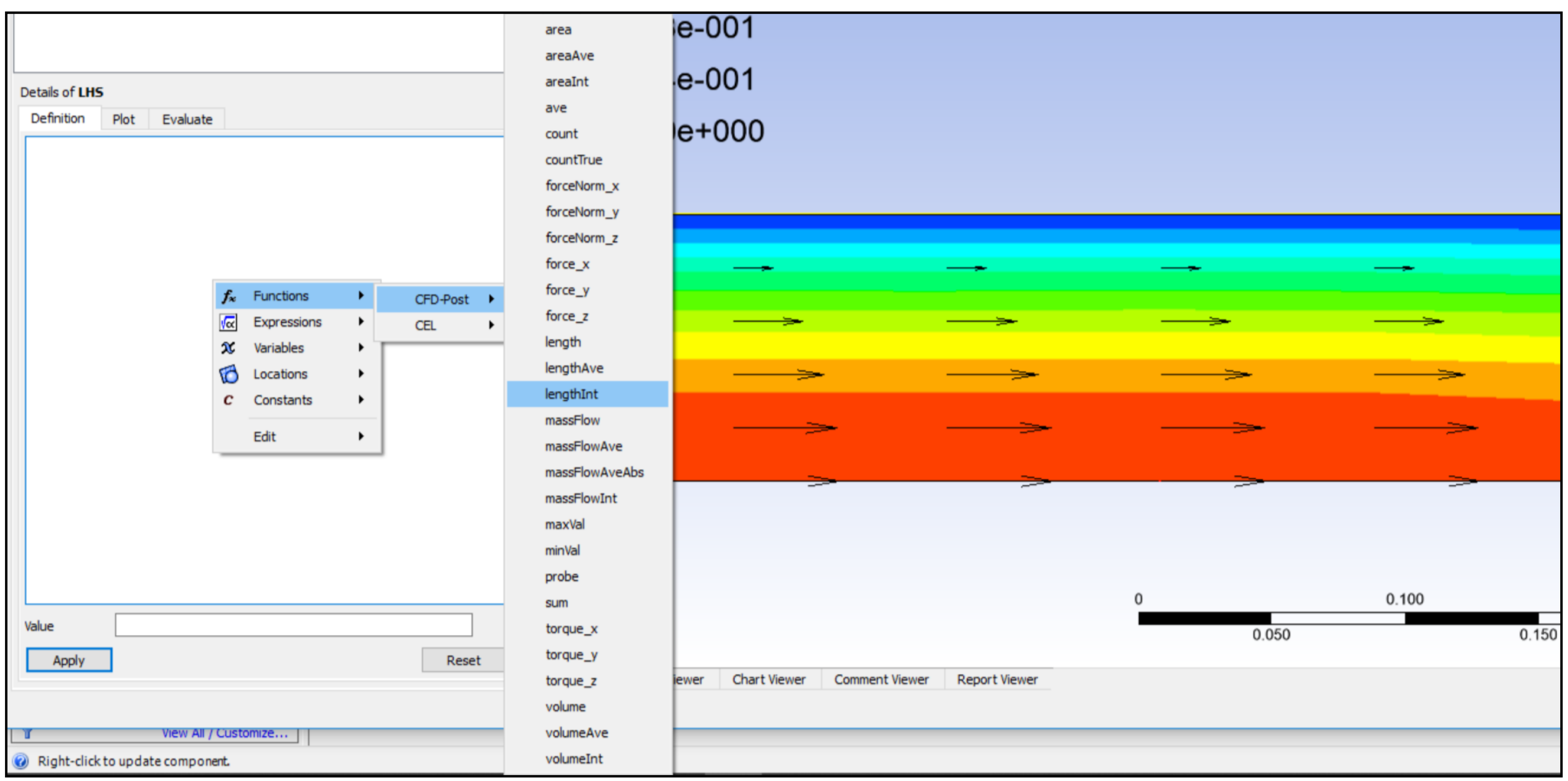
R-Click on Details of LHS and select Functions/CFD-Post/**LenghtInt**

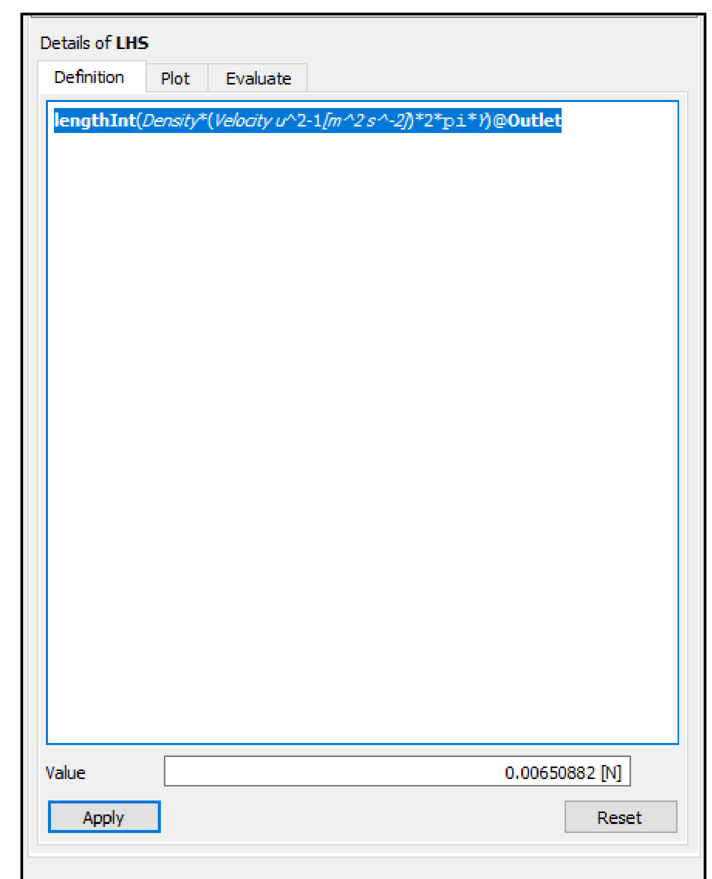
R-Click inside the bracket lengthInt**(here)** and <u>apply the formula as below</u>

Define: “ **lengthInt**(*Density**(*Velocity u*^2-1*[m^2 s^-2]*)*2*pi**Y*)@**Outlet** “

Note that the *Density*, *Velocity u and Y* are Variables and it should be selected by: R-Click/select **Variables/…**

Note that the **Inlet/Outlet** are Locations and it should be selected by: R-Click/select **Locations/Inlet or Outlet**





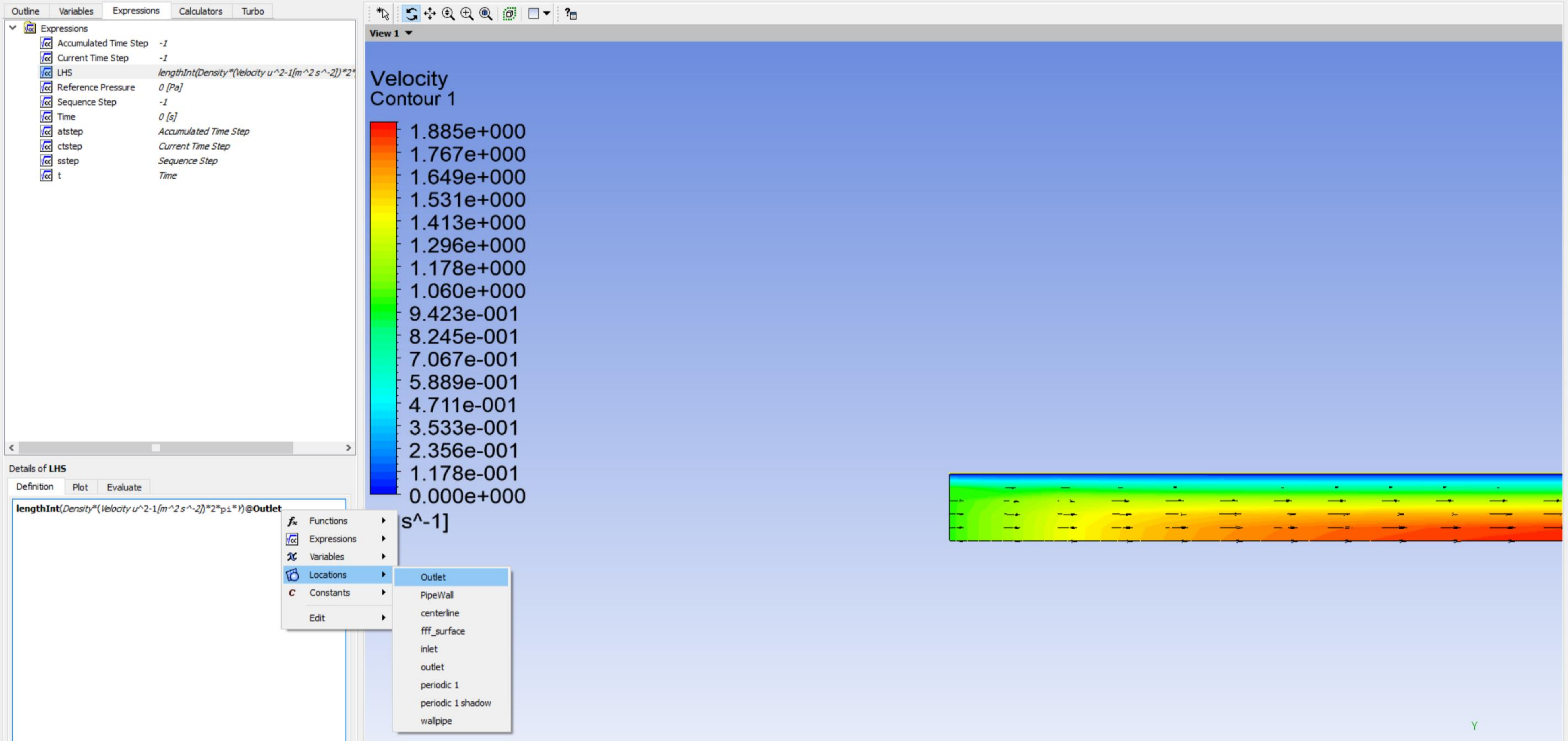
CFD-Post (Objective: Pressure Force)

Location/Line, Change the Name to **Inlet**, then Ok

Set the Point 1 to **(0)(0)(0)** and the Point 2 to **(0)(0.1)(0)**, then Apply

Click on **Expressions** tab, R-Click on Expressions, then click on New

Change the Name to **RHS**, then Ok



CFD-Post (Objective: Expressions, Momentum Balance)

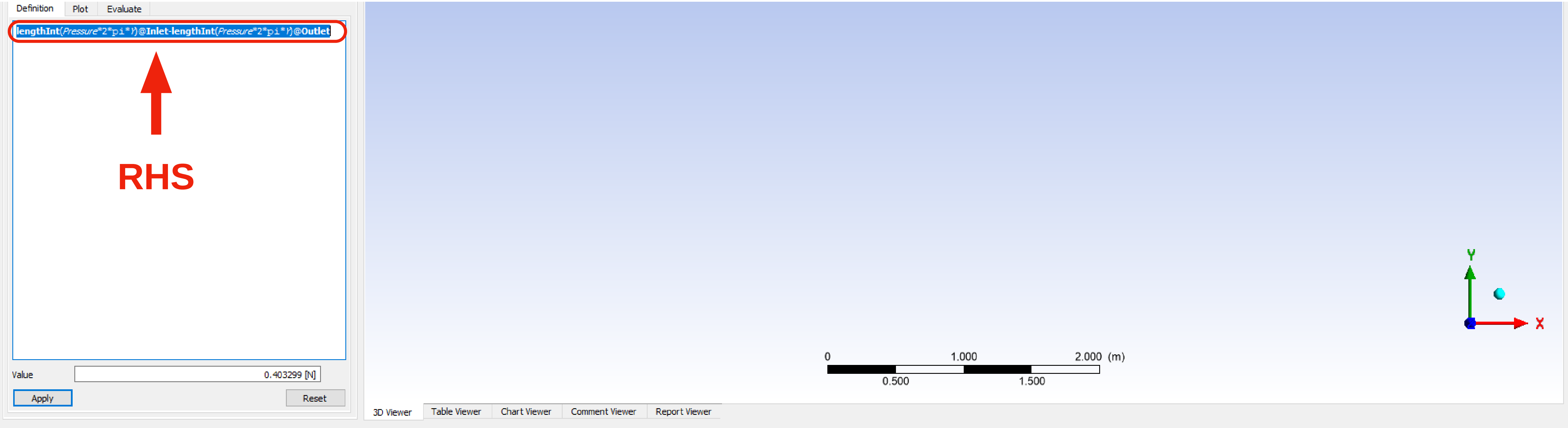
R-Click on Details of RHS and select Functions/CFD-Post/**LenghtInt**

Define:

lengthInt(*Pressure**2*pi**Y*)**@Inlet - lengthInt**(*Pressure**2*pi**Y*)**@Outlet**

Note that the *Pressure and Y* are Variables and it should be selected by: R-Click/select **Variables/…**

Note that the **Inlet/Outlet** are Locations and it should be selected by: R-Click/select **Locations/Inlet or Outlet**



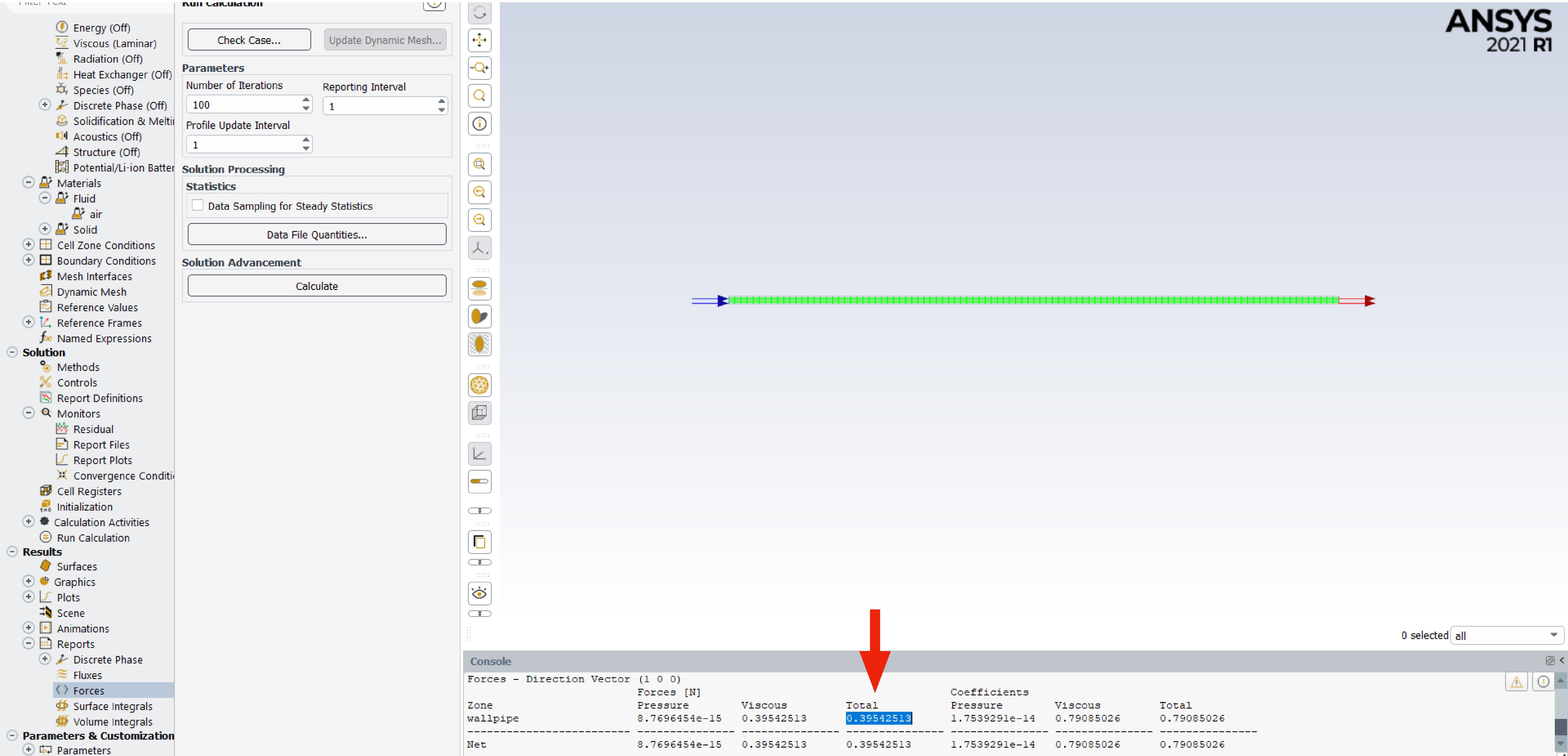
CFD-Post (Objective: Drag Force)

Double click on **Solution** from Workbench to open Fluent

At the Tree Menu select Results/Reports/**Forces** (D-click on Forces)

Click on **Print**, Results will be calculated and appeared at Console box

Write down the Total Force as shown in picture and close Fluent



CFD-Post (Objective: Drag Force)

Double click on **Results** from Workbench

Click on **Expressions** tab, R-Click on Expressions, then click on New

Change the Name to **Fshear**, then Ok

Paste (write) the copied number and define a unit [N], then **Apply**

CFD-Post (Objective: Momentum Balance)

Click on **Expressions** tab, R-Click on Expressions, then click on New

Change the Name to **MomBalance**, then Ok

Add the copied number for Total Force and define a unit [N], then Apply

Define: **(RHS-LHS-Fshear)/Fshear*100**

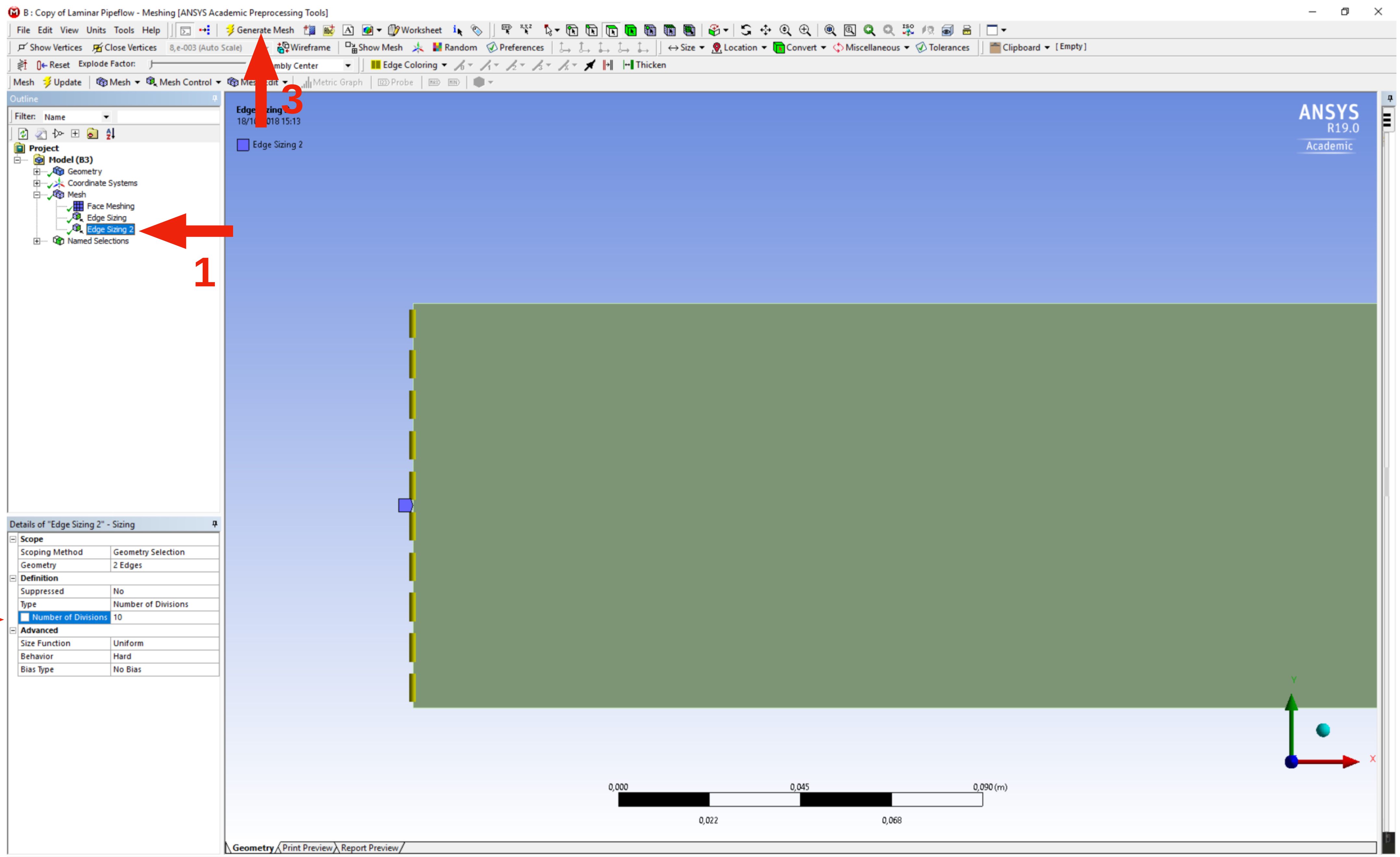
CFD-Post (Objective: Results Optimization - Mesh Resolution)

From Workbench, R-Click on Mesh and Click on Duplicate

Double click on new duplicated Mesh

From Outline Menu Click on Model/Mesh/Edge Sizing 2

Increase the Number of Divisions to **10**, then Click on **Generate Mesh**



CFD-Post (Objective: Results Optimization)

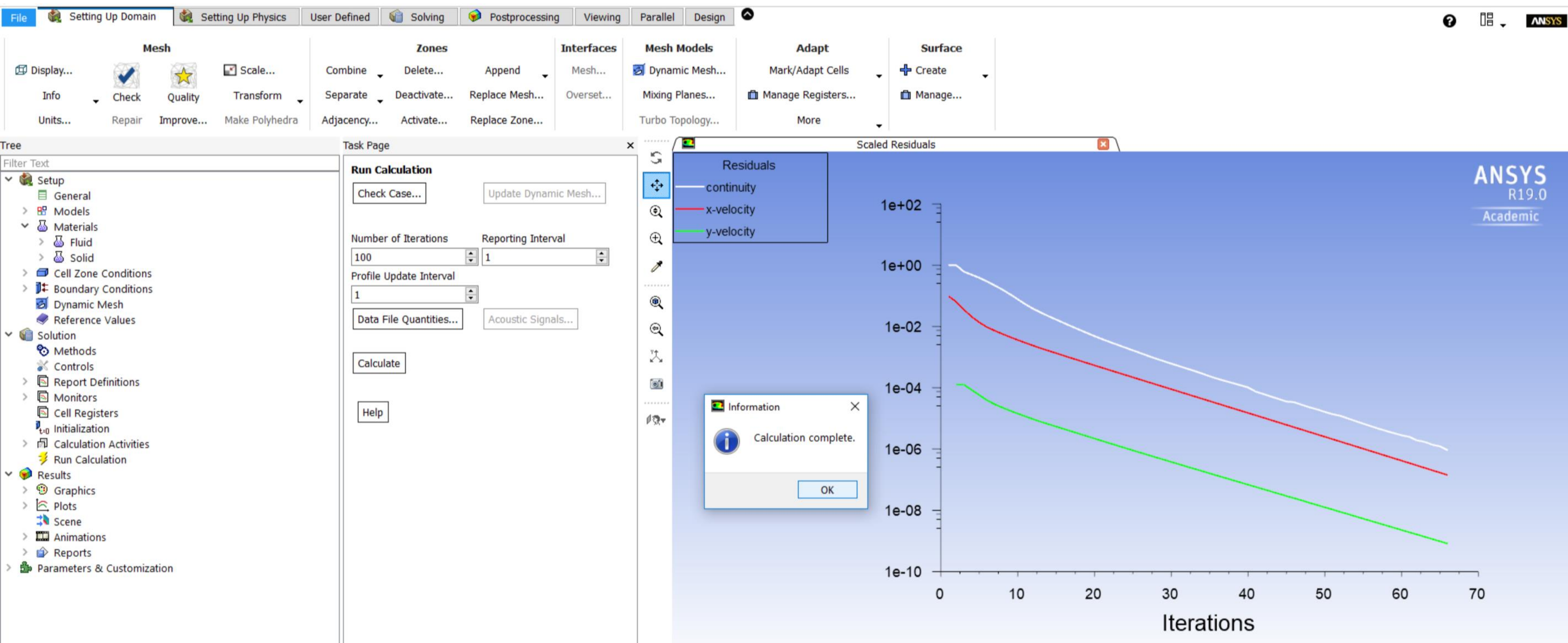
From Workbench, R-Click on duplicated Mesh and Click on **Update**

D-Click on Setup

At the Tree Menu double click on Solution/**Initialization,**

Change the Compute from to **Inlet** then Click on the **Initialize** icon

At the Tree Menu double click on Solution/**Run Calculation** then **Calculate**

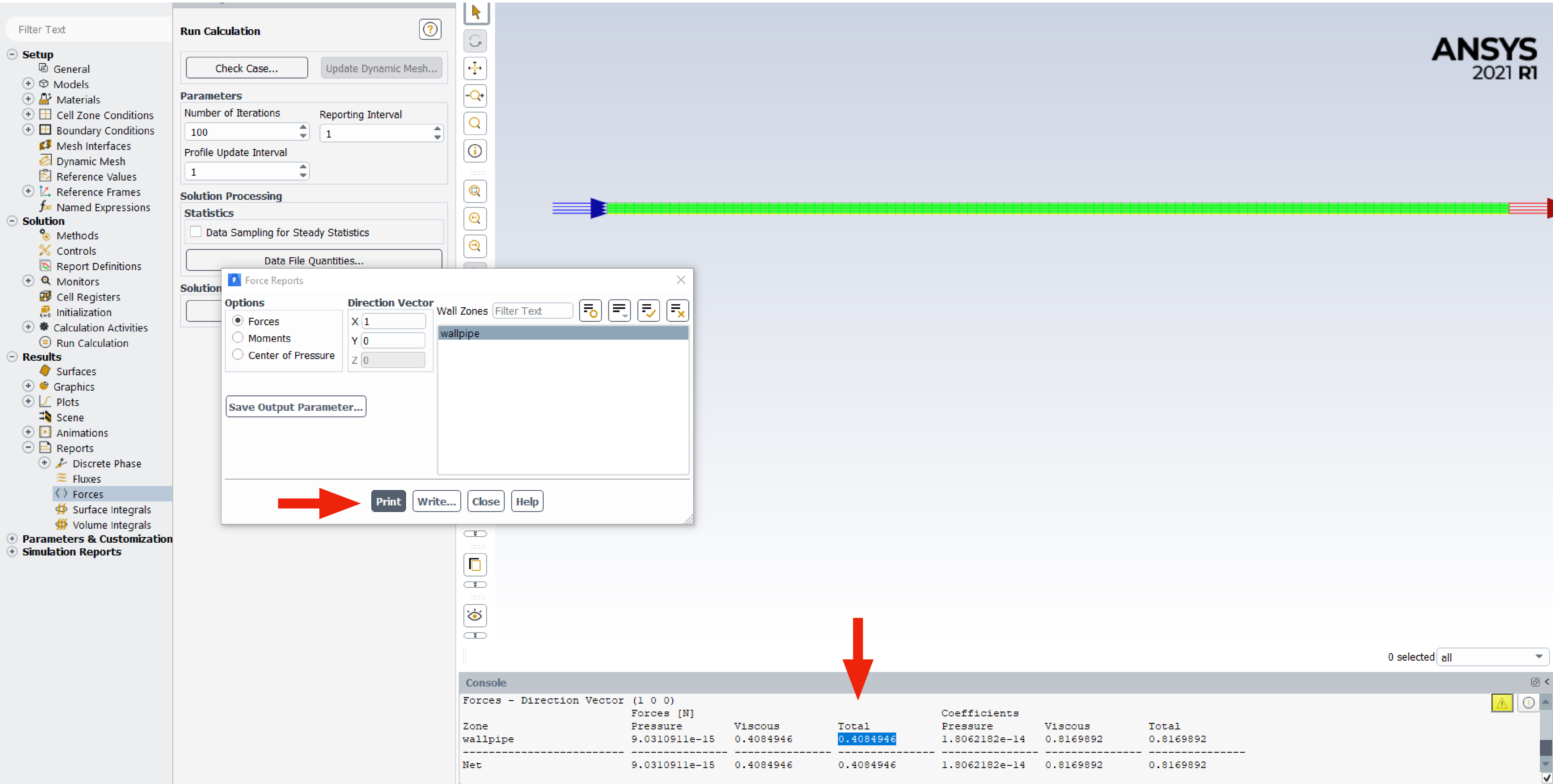


CFD-Post (Objective: Drag Coefficient)

At the Tree Menu select Results/Reports/**Forces** (D-click on Fluxes)

Click on **Print**, Results will be calculated and appeared at Console box

Write down the Total Force as shown in picture and close Fluent

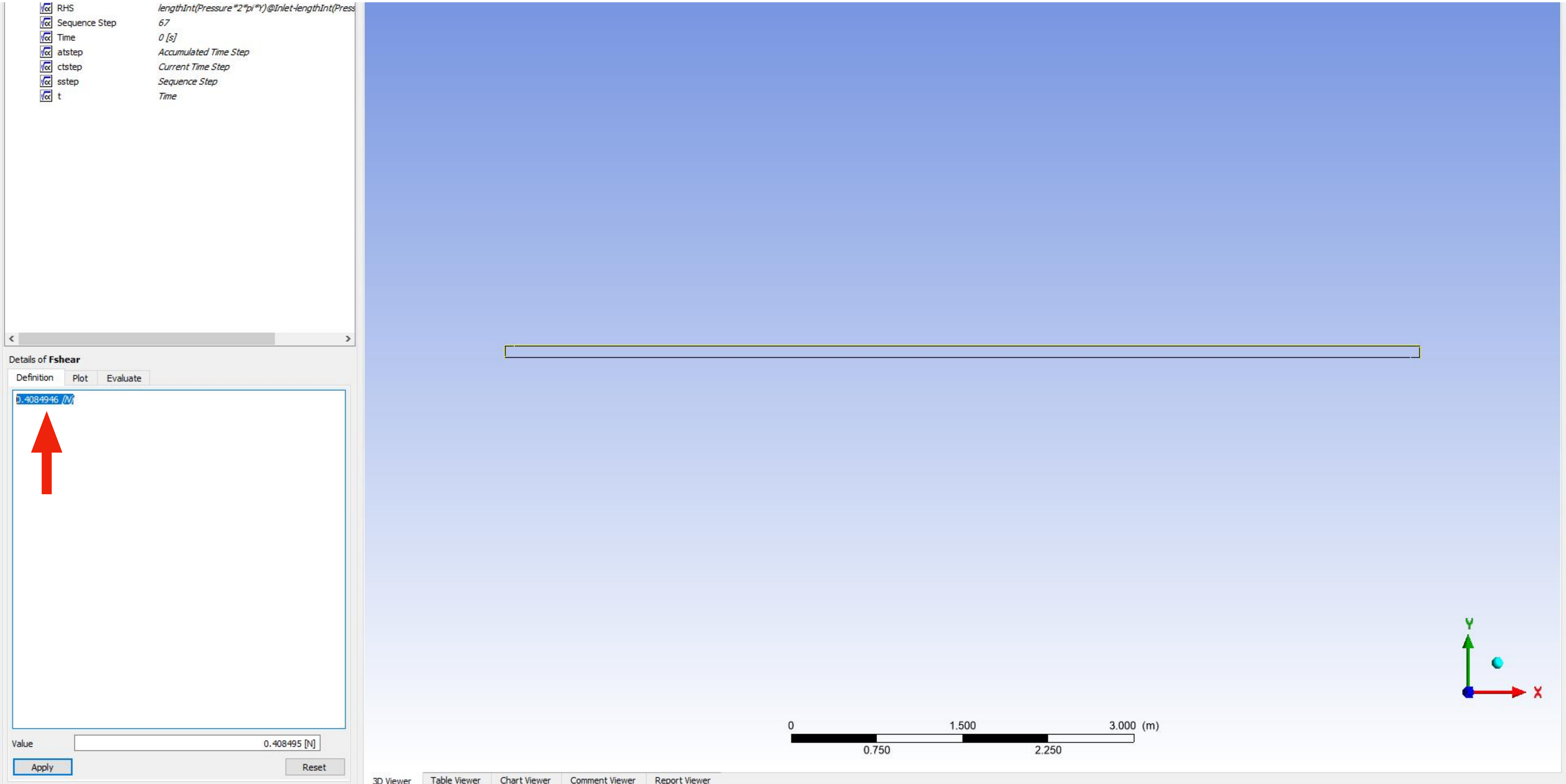


CFD-Post (Objective: Drag Force)

Double click on **Results** from duplicated Workbench

Click on **Expressions** tab, Click on **Fshear** and Modify the amount

Paste (write) the copied number and define a unit [N], then Apply

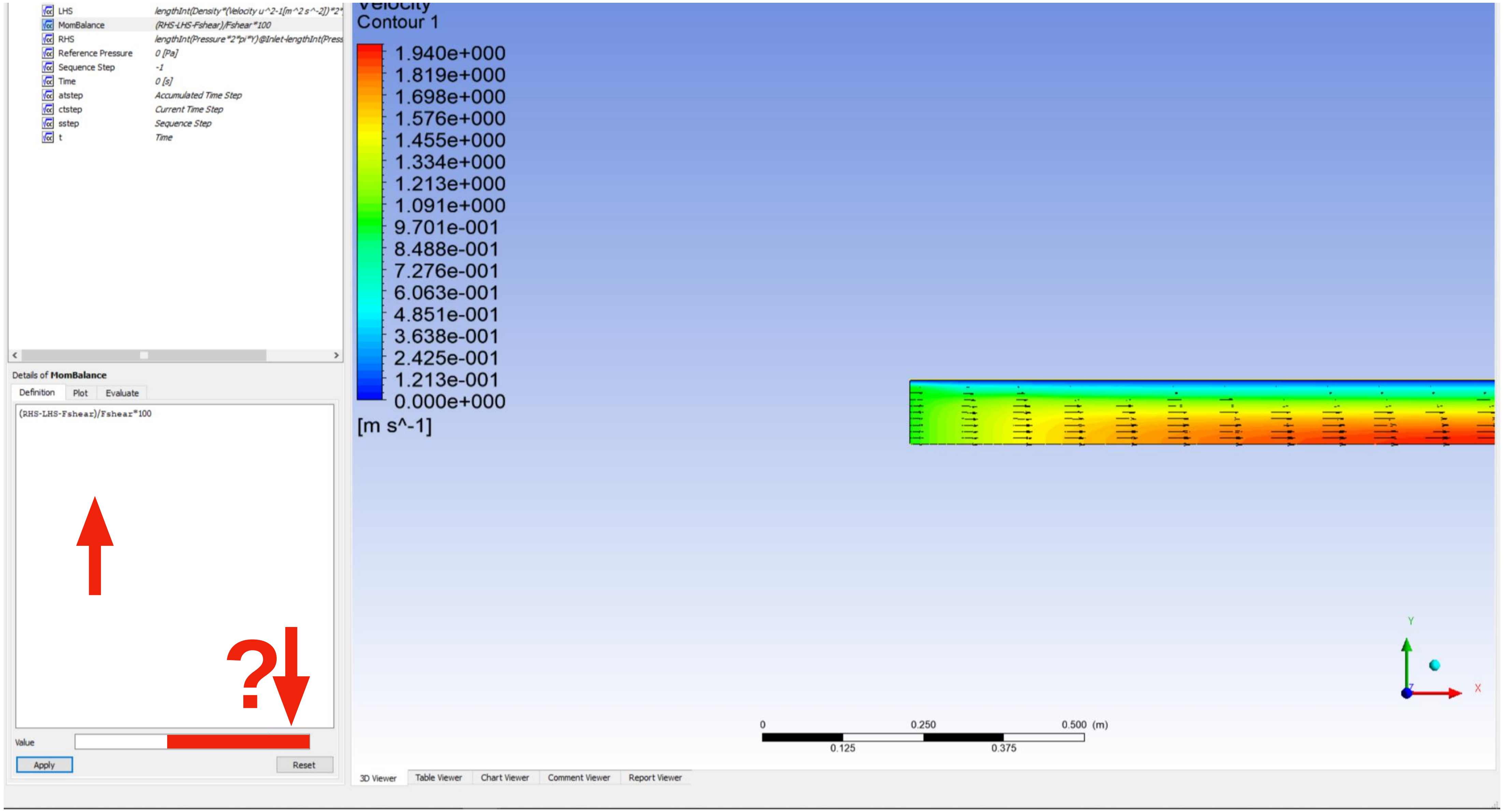


CFD-Post (Objective: Drag Force)

Click on **MomBalance**, then Apply

Check the differences of **MomBalance** in two cases

Submit the screen capture including the value of MomBalance



Assignments

Submit the workbench case file and folder, and the screen capture (as shown in the previous slide, including the value of MomBalance) within a .zip. The submitted .zip must be named as follow:

<StudentLastName>_<StudentFirstName>_Tutorial.zip