

Environmental Transport Phenomena

Project – Part I: Tutorial

Laminar Pipe Flow

Mohammad Jamshidmofid

mohammad.jamshidmofid@epfl.ch

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.

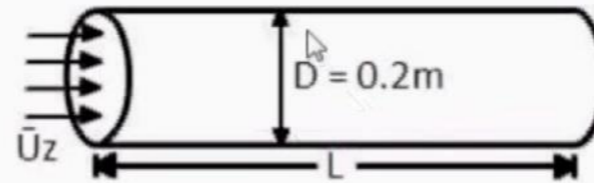
Assignment

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/m³ 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).

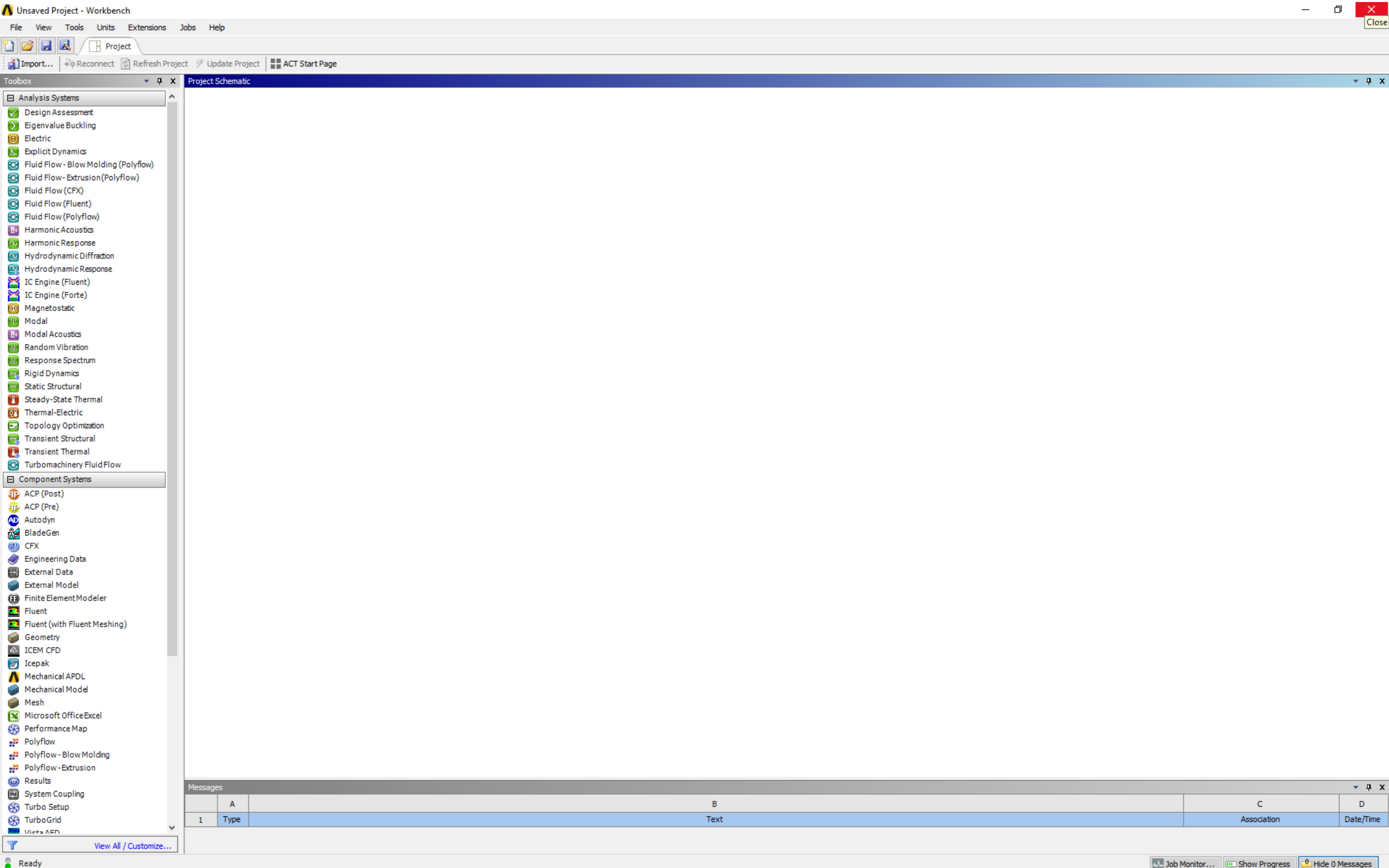
Summary of the assignment

Laminar Pipeflow



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

Run Workbench: (start - Workbench 2021 R1)



Unsaved Project - Workbench

File View Tools Units Extensions Jobs Help

Project

Import... Reconnect Refresh Project Update Project ACT Start Page

Toolbox

- Analysis Systems
 - Design Assessment
 - Eigenvalue Buckling
 - Electric
 - Explicit Dynamics
 - Fluid Flow - Blow Molding (Polyflow)
 - Fluid Flow - Extrusion (Polyflow)
 - Fluid Flow (CFX)
 - Fluid Flow (Fluent)
 - Fluid Flow (Polyflow)
 - Harmonic Acoustics
 - Harmonic Response
 - Hydrodynamic Diffraction
 - Hydrodynamic Response
 - IC Engine (Fluent)
 - IC Engine (Forte)
 - Magnetostatic
 - Modal
 - Modal Acoustics
 - Random Vibration
 - Response Spectrum
 - Rigid Dynamics
 - Static Structural
 - Steady-State Thermal
 - Thermal-Electric
 - Topology Optimization
 - Transient Structural
 - Transient Thermal
 - Turbomachinery Fluid Flow
- Component Systems
 - ACP (Post)
 - ACP (Pre)
 - Auto dyn
 - BladeGen
 - CFX
 - Engineering Data
 - External Data
 - External Model
 - Finite Element Modeler
 - Fluent
 - Fluent (with Fluent Meshing)
 - Geometry
 - ICEM CFD
 - Icepak
 - Mechanical APDL
 - Mechanical Model
 - Mesh
 - Microsoft Office Excel
 - Performance Map
 - Polyflow
 - Polyflow - Blow Molding
 - Polyflow - Extrusion
 - Results
 - System Coupling
 - Turbo Setup
 - TurboGrid
 - Vieta AFD

Project Schematic

Messages

	A	B	C	D
1	Type	Text	Association	Date/Time

Ready Job Monitor... Show Progress Hide 0 Messages

Drag the Fluid Flow icon to the Project Schematic window

The screenshot displays the ANSYS Workbench interface. On the left, the 'Toolbox' is open, showing a list of analysis systems. A red arrow points to the 'Fluid Flow (Fluent)' icon. The 'Project Schematic' window on the right shows a project tree with the following items: 1. Fluid Flow (Fluent), 2. Geometry, 3. Mesh, 4. Setup, 5. Solution, and 6. Results. The 'Fluid Flow (Fluent)' icon is currently being dragged from the toolbox to the 'Fluid Flow (Fluent)' item in the project tree.

Messages

	A	B	C	D
1	Type	Text	Association	Date/Time

Ready Job Monitor... Show Progress Hide 0 Messages

Run Geometry (SpaceClaim Direct Modeler)

The screenshot displays the ANSYS Workbench interface. The main window is titled "Project Schematic" and shows a project tree with the following items:

- 1 Fluid Flow (Fluent) - highlighted with a red arrow
- 2 Geometry
- 3 Mesh
- 4 Setup
- 5 Solution
- 6 Results

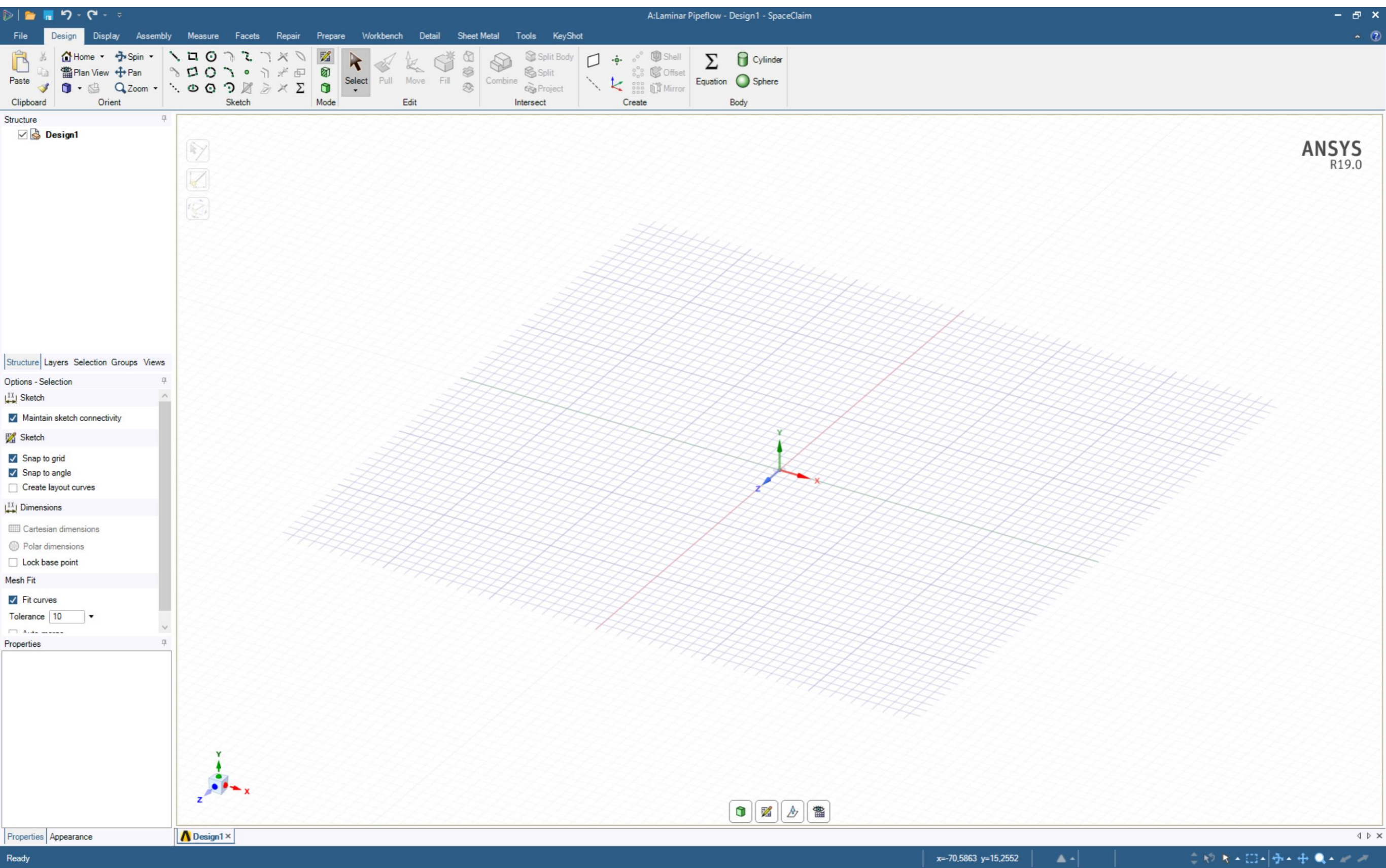
Below the project tree, the "laminar Pipeflow" model is visible. The left sidebar contains the "Toolbox" with various analysis systems and component systems. The "Fluid Flow (Fluent)" option is selected under the "Analysis Systems" category.

The bottom of the interface shows a "Messages" panel with a table containing the following data:

	A	B	C	D
1	Type	Text	Association	Date/Time

The status bar at the bottom indicates "Ready" and includes buttons for "Job Monitor...", "Show Progress", and "Hide 0 Messages".

1- SpaceClaim Direct Modeler



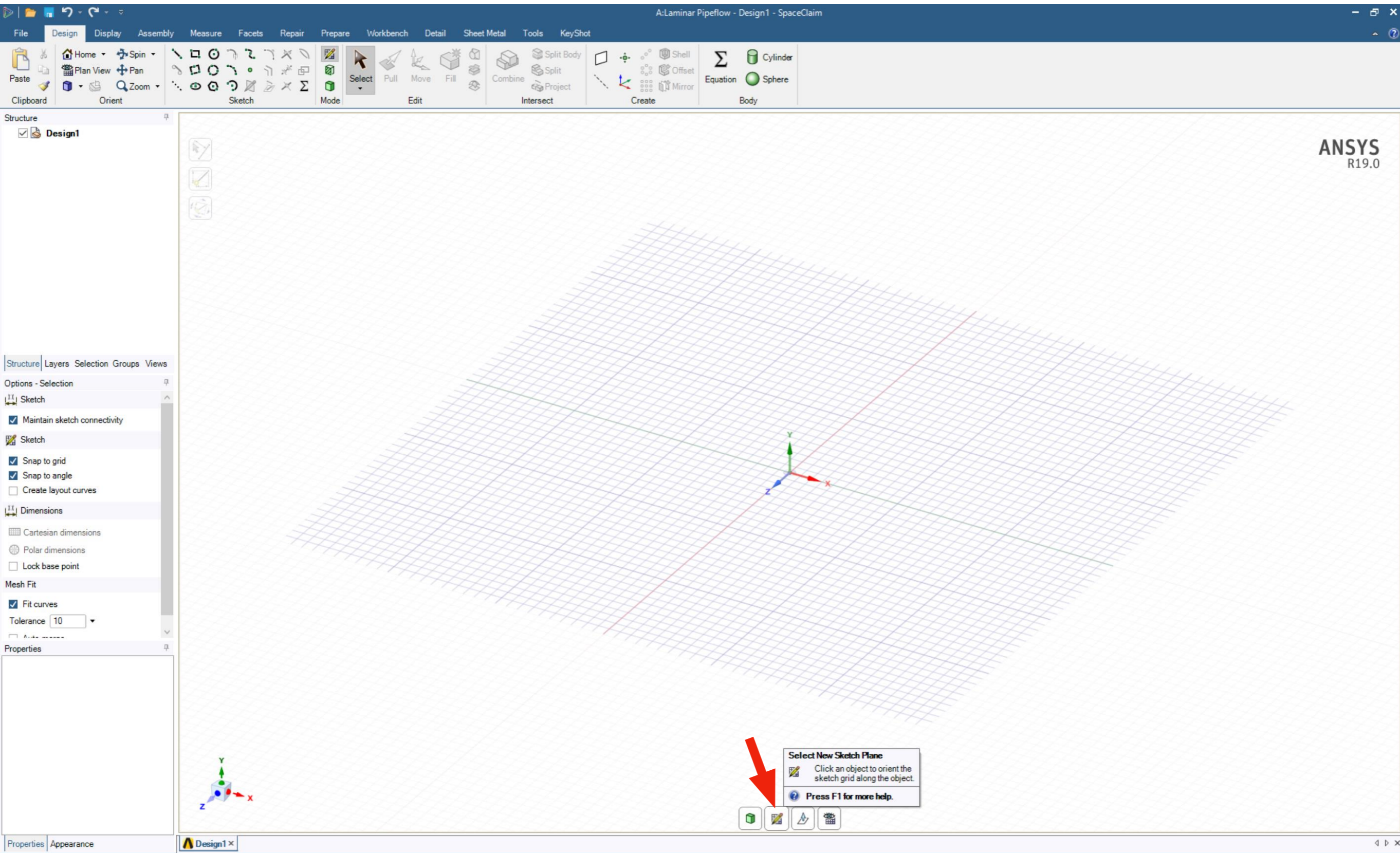
Change units in SpaceClaim

File/SpaceClaim Options/Units

The screenshot displays the ANSYS SpaceClaim R19.0 interface. The 'SpaceClaim Options' dialog box is open, showing the 'Units' section. The 'Units settings for:' dropdown is set to 'This Document'. The 'Length' dropdown menu is open, showing options: Nanometers, Micrometers, Millimeters, Centimeters, and Meters. A red arrow labeled '2' points to the 'Meters' option. The 'OK' button at the bottom right of the dialog is also highlighted with a red arrow labeled '3'. The 'File' menu in the top-left corner is highlighted with a red arrow labeled '1'. The 'Structure' tree on the left shows 'Design1' selected. The 'Properties' panel at the bottom left shows 'Appearance' selected. The status bar at the bottom indicates 'Ready' and 'x=-18,5155 y=6,4564'.

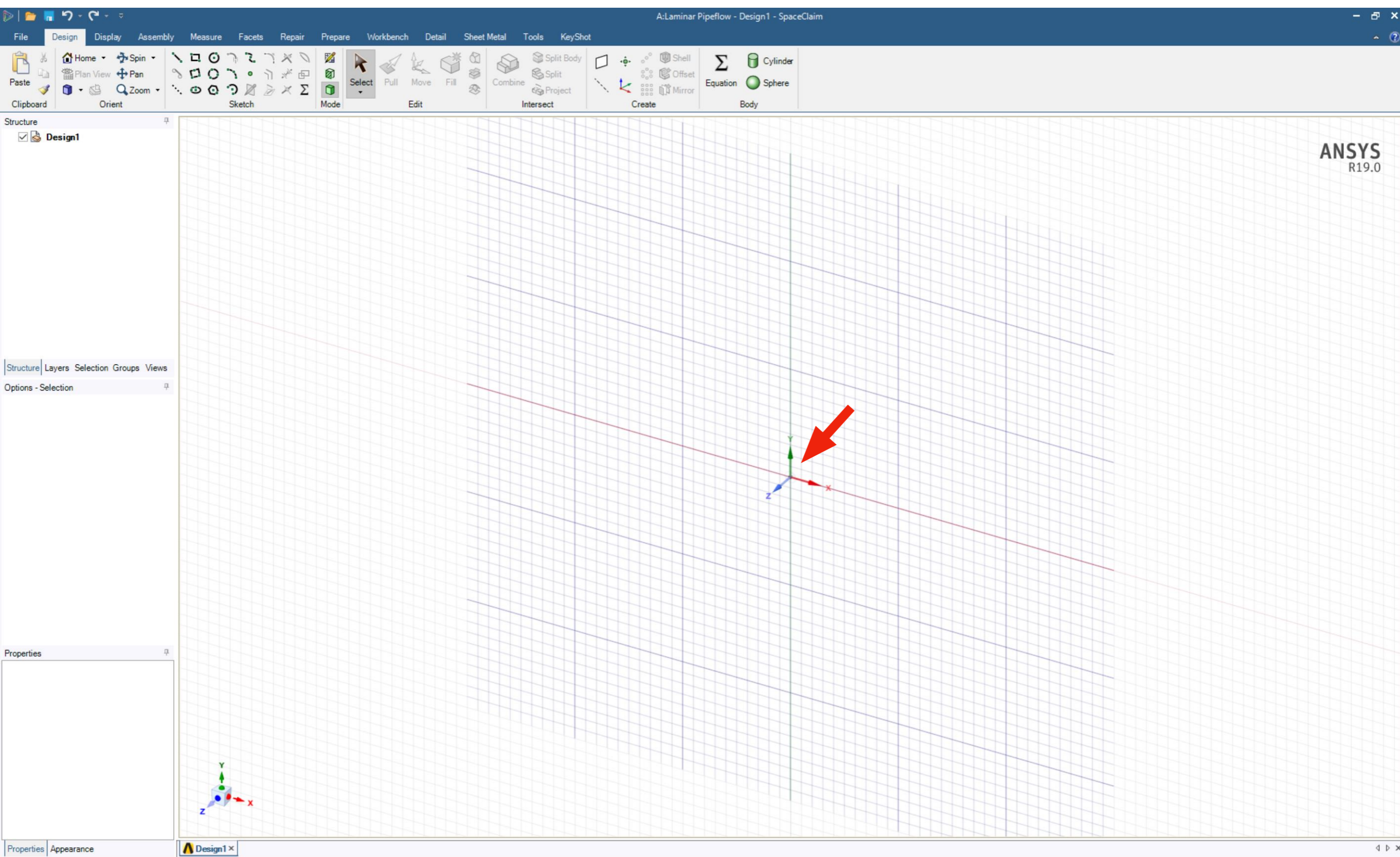
Modify Sketch Plane

Select New Sketch Plane



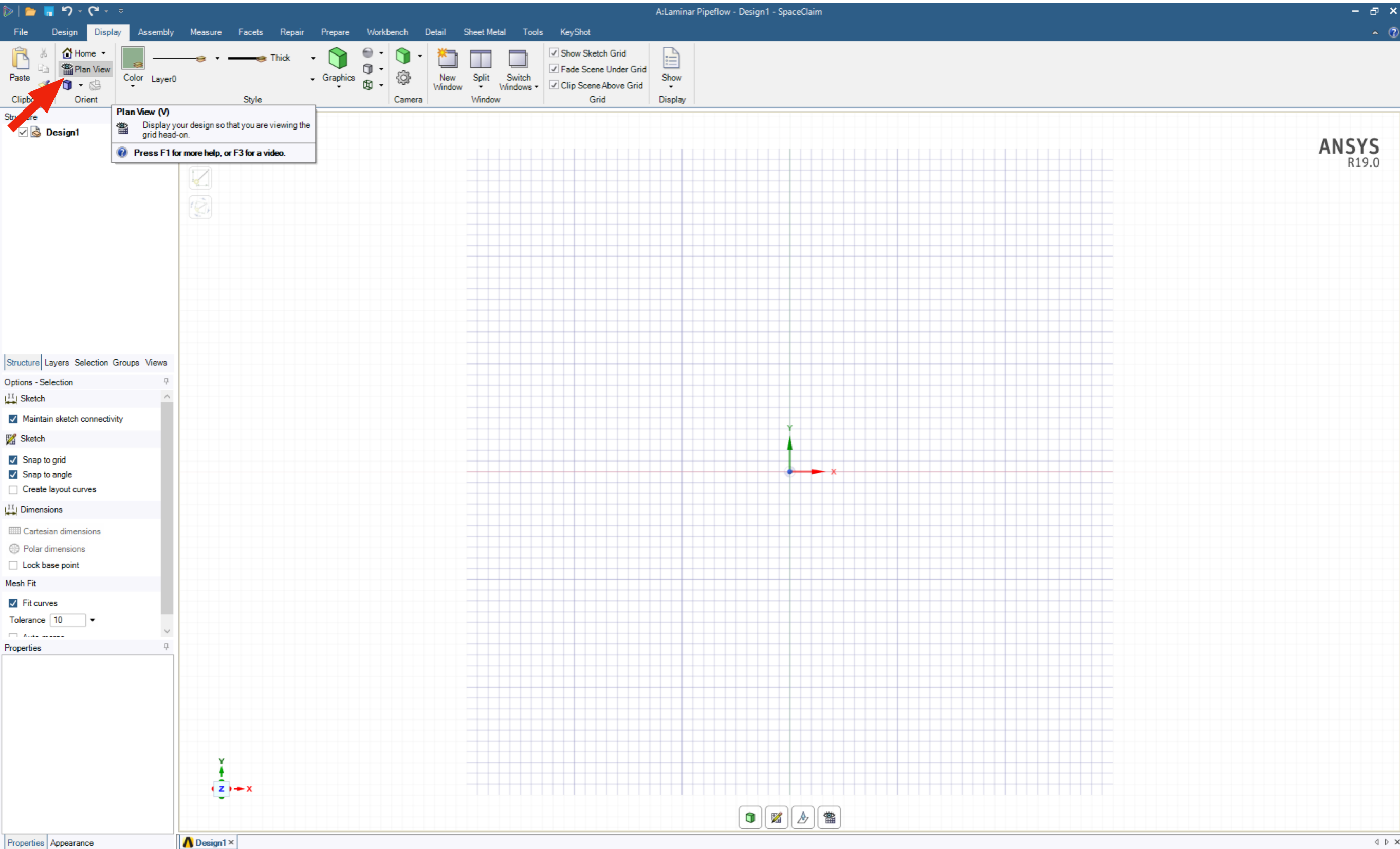
Modify Sketch Plane

Move the pointer on XY Plane / L-Click



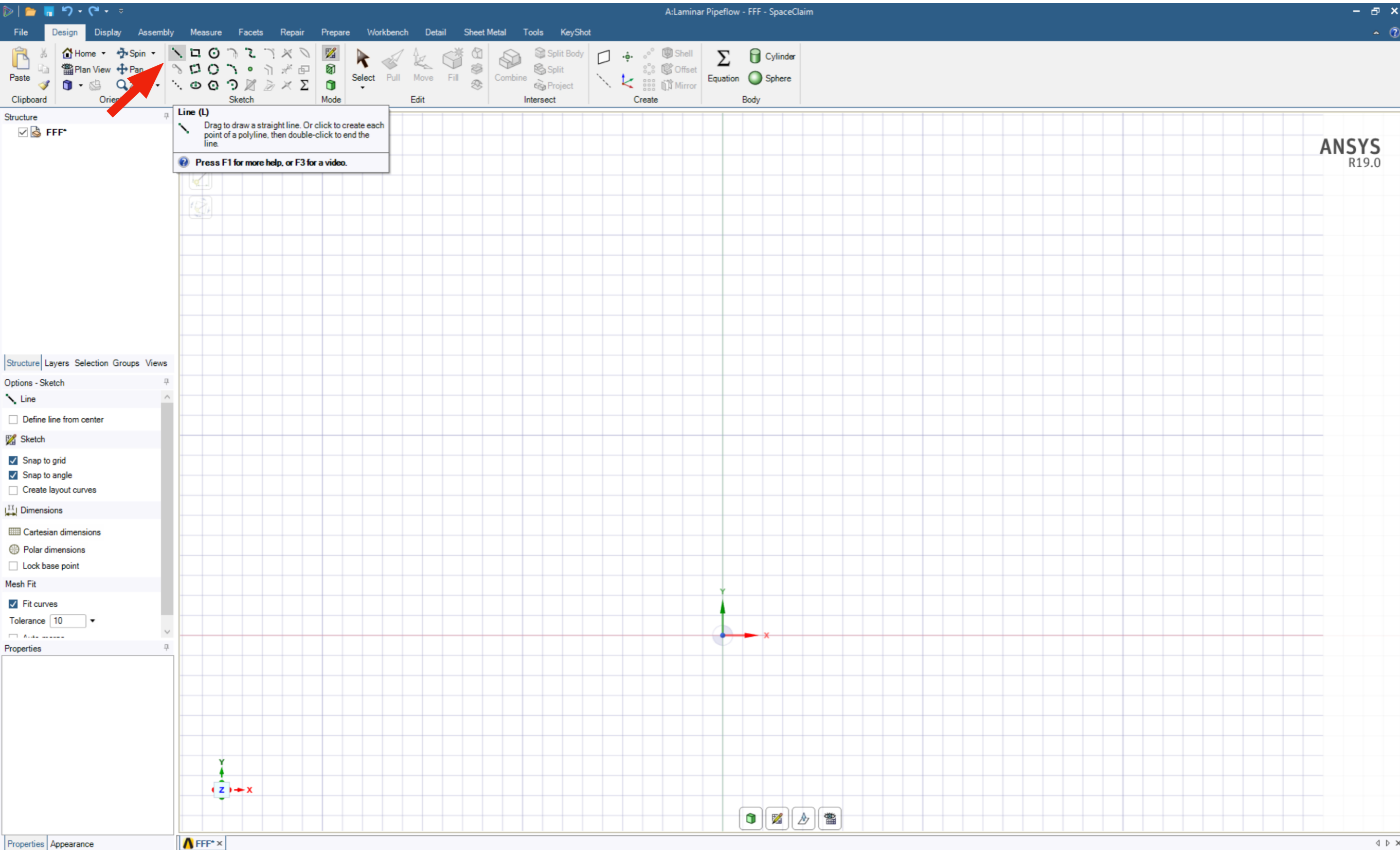
Plane View

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

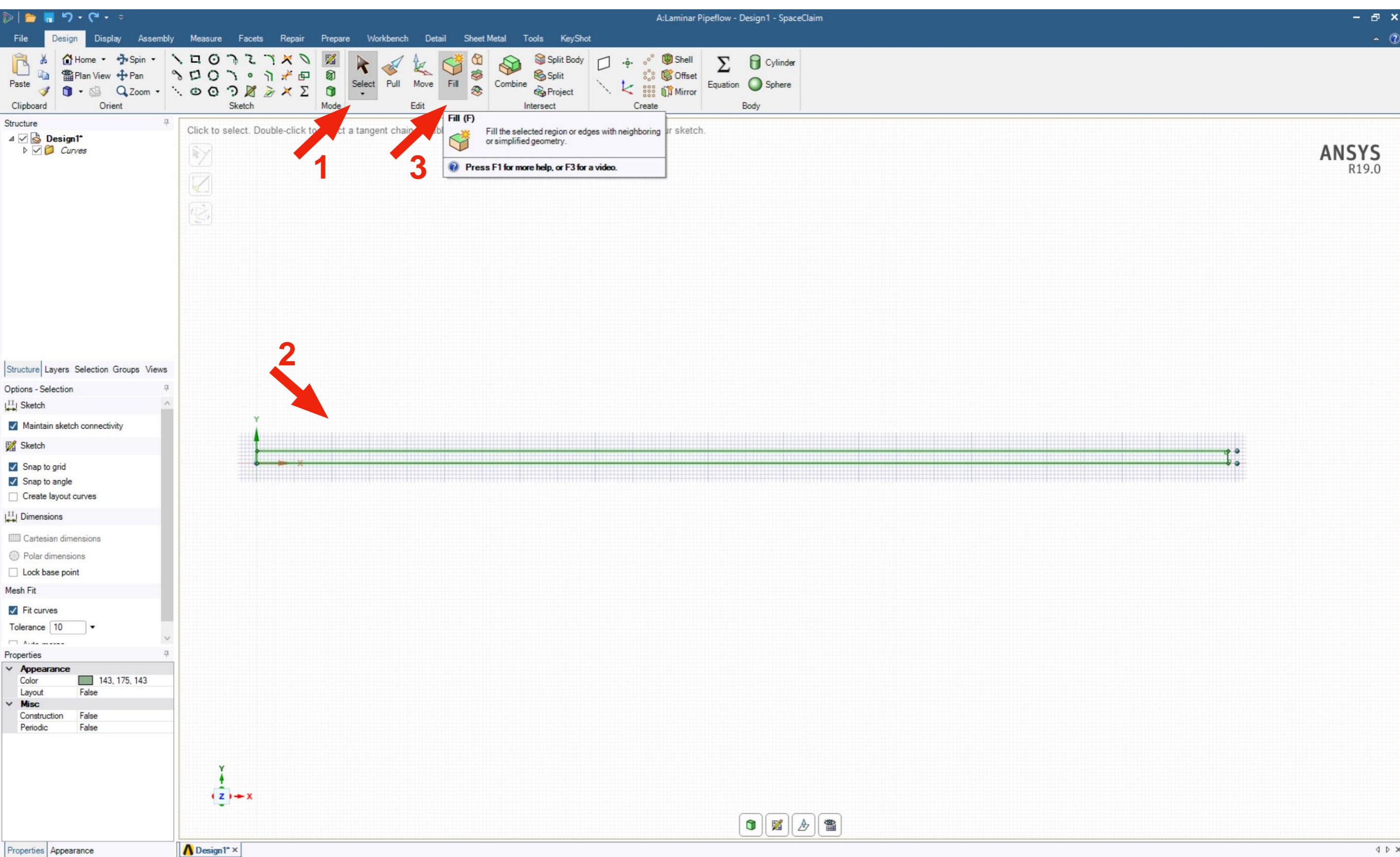


Be careful in drawing straight lines!

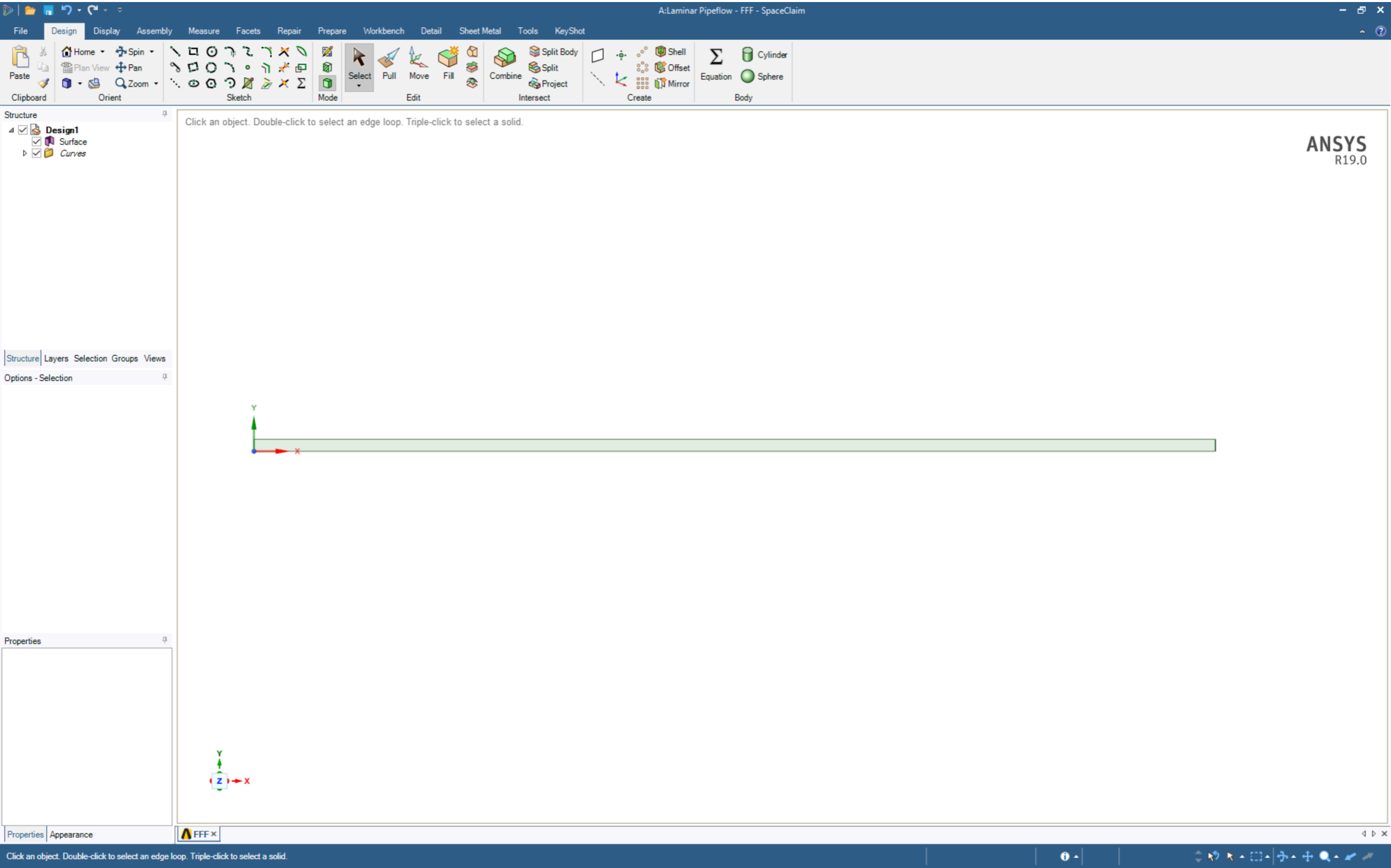
The screenshot displays the ANSYS R19.0 software interface. The top toolbar includes various design tools such as Home, Spin, Plan View, Pan, Zoom, Orient, Sketch, Mode, Edit, Intersect, Create, and Body. The left sidebar shows the 'Structure' tree with 'FFF*' and 'Curves' folders, and the 'Options - Sketch' panel with settings for 'Line', 'Sketch', and 'Dimensions'. The main workspace shows a sketch of a horizontal line with a dimension of 0.1 m. A red arrow points to the '0.1 m' dimension label. The coordinate system (X, Y, Z) is visible at the bottom left. The status bar at the bottom indicates 'Snapping to Perpendicular to Endpoint (2), Parallel to Grid, Endpoint, Perpendicular to Curve (2), Parallel to Curve, Dimension Value' and 'x=0,0000 y=0,0000'.

Fill the selected edges to create a surface

Design/select/select all the edges/Fill

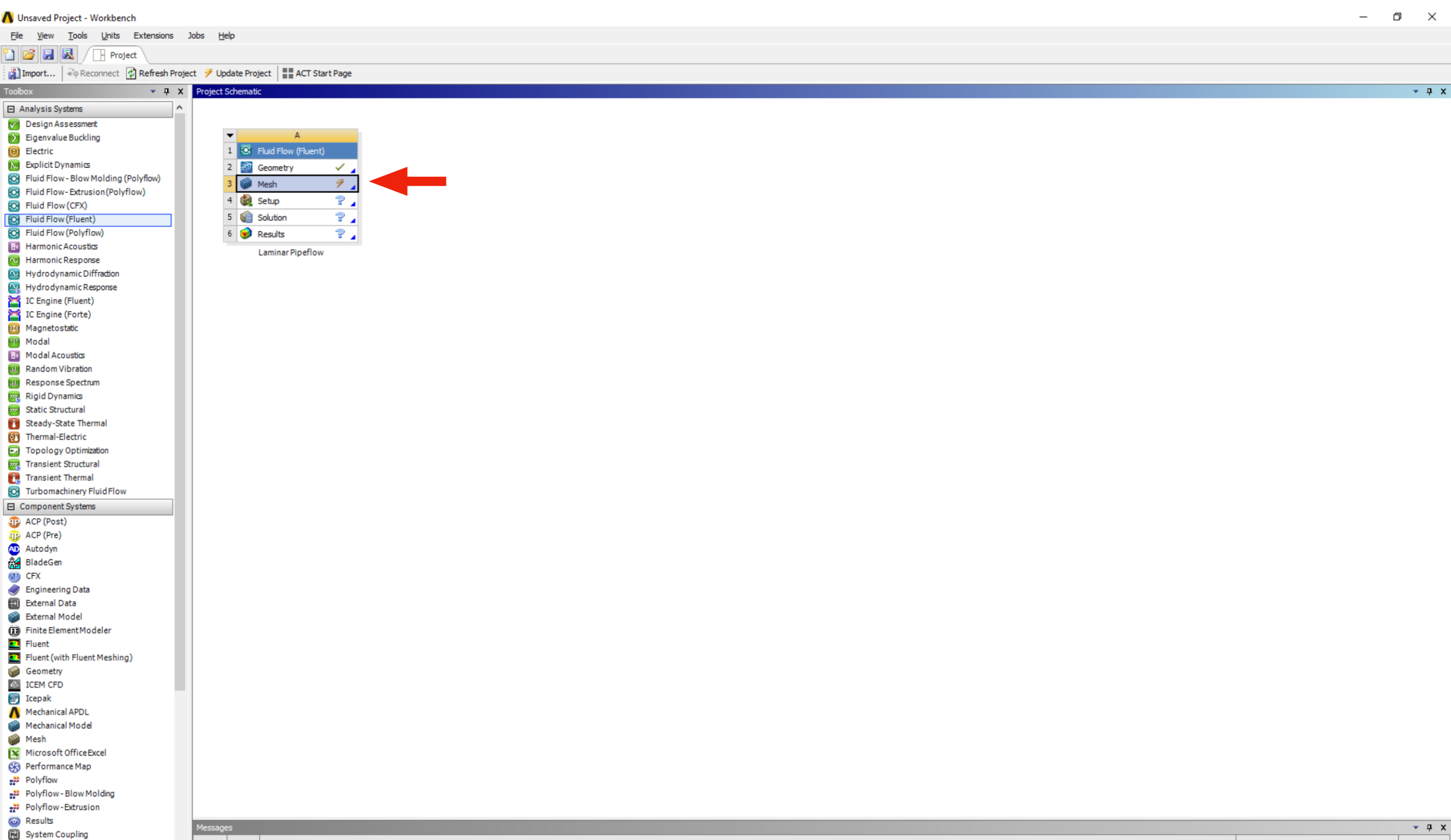


SpaceClaim Results



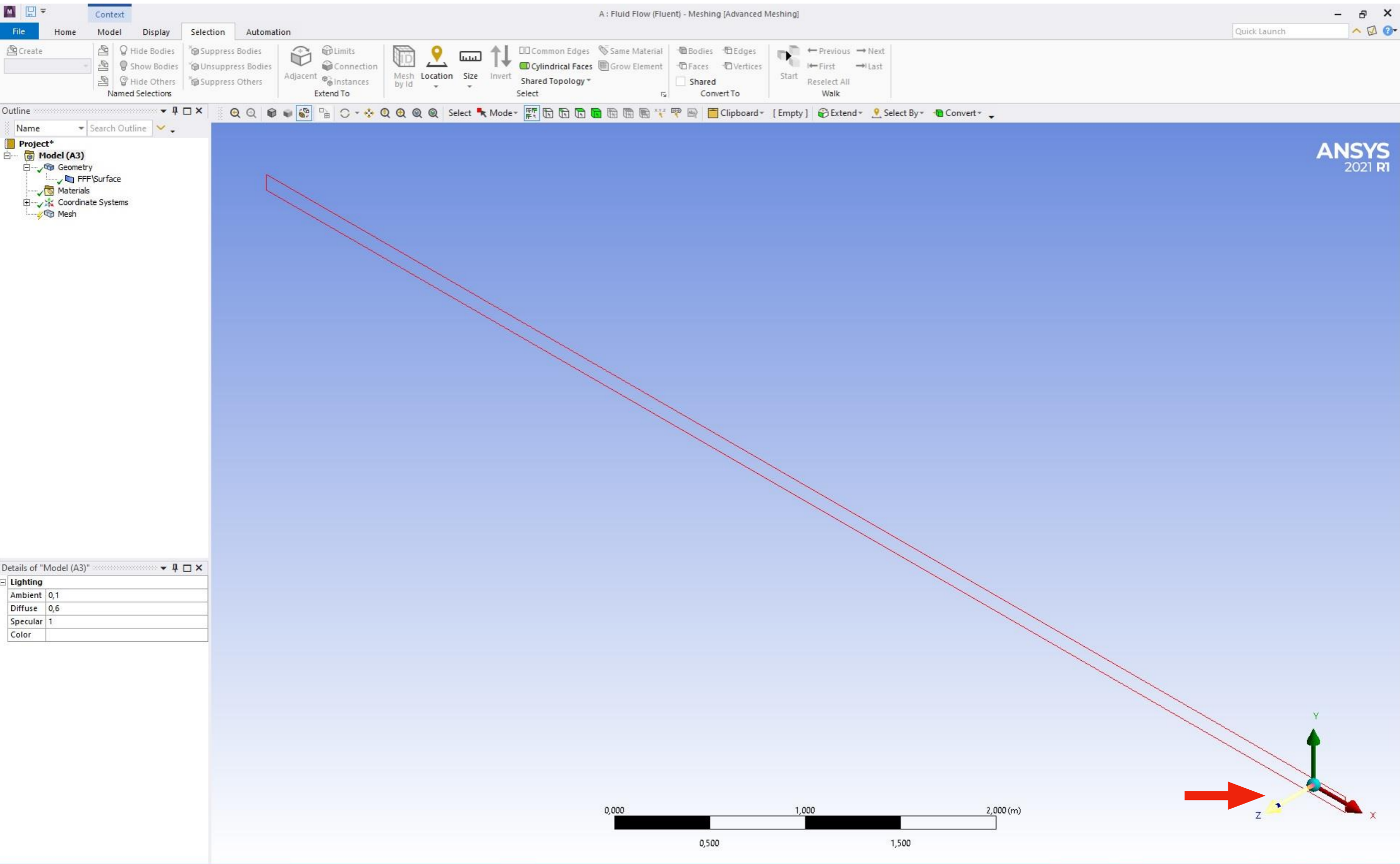
2- Run Mesh (From Workbench)

Control Green checkmark of Geometry/Save the project
Run Mesh from Workbench



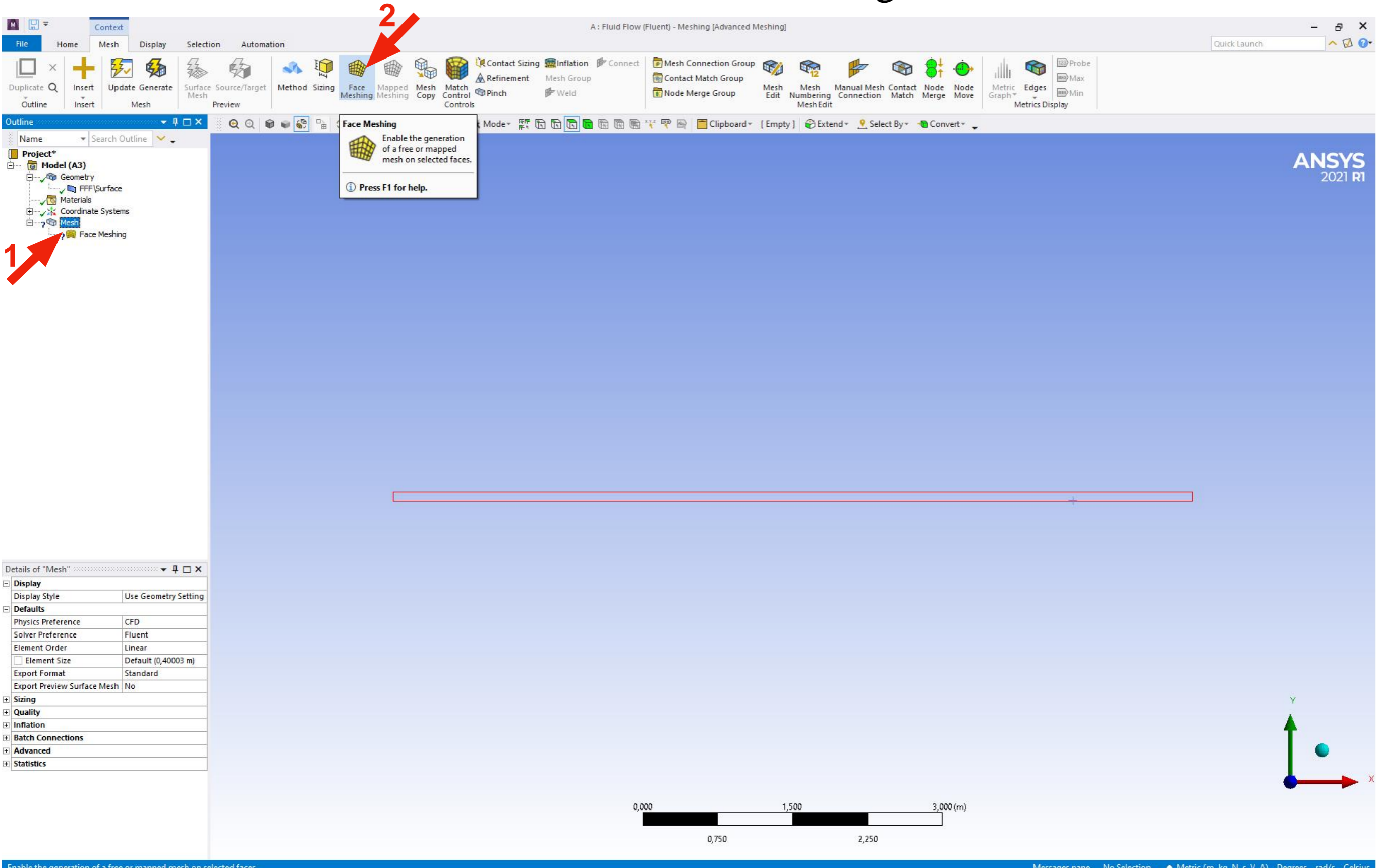
Plane View

L-Click on the z-axis



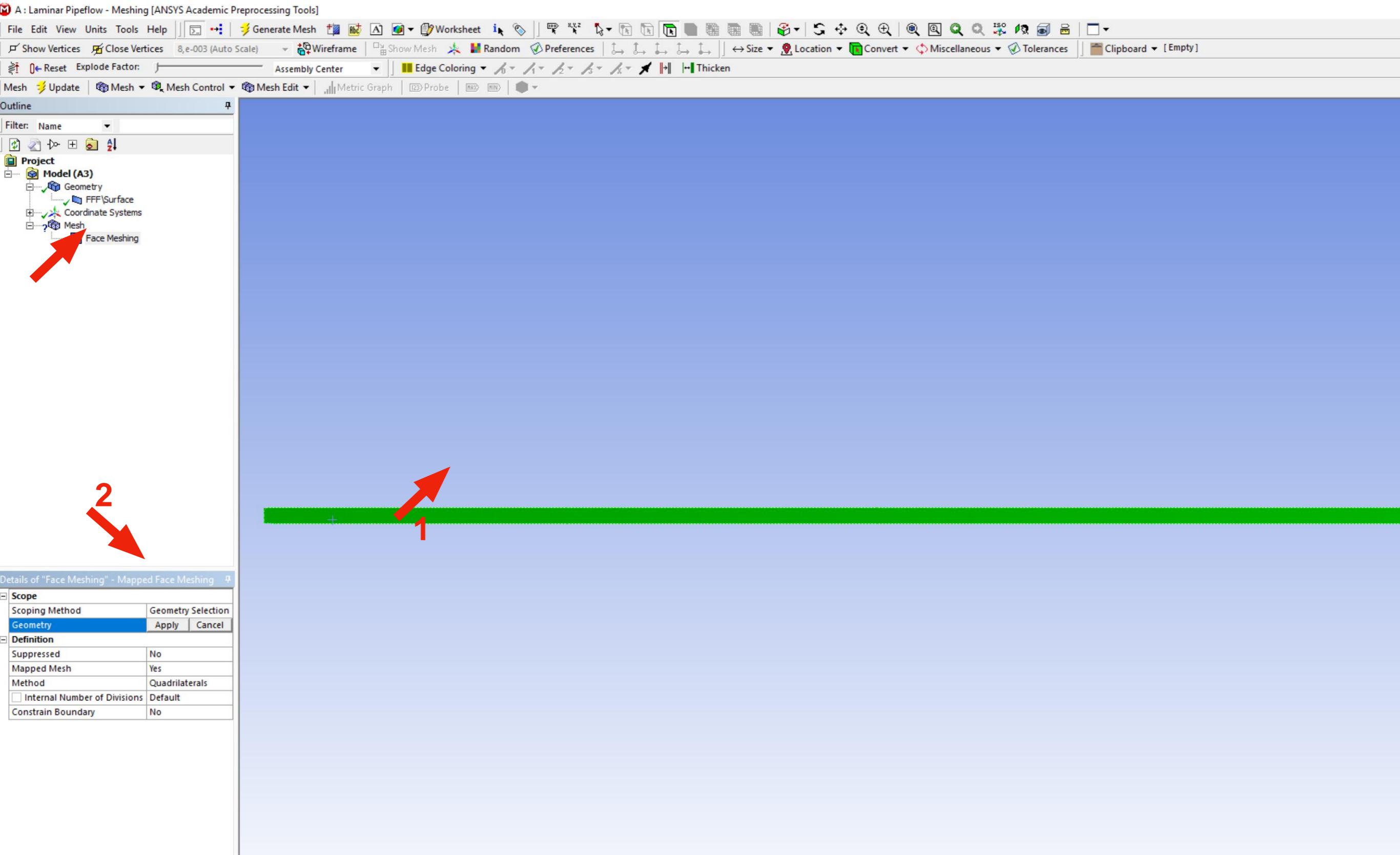
Face meshing

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

The screenshot displays the ANSYS 2021 R1 Meshing software interface. The main window shows a 3D model of a cylinder with a mesh applied. A red arrow points to the 'Mesh' icon in the Outline window on the left. Another red arrow points to the 'Sizing' icon in the top toolbar. A tooltip for the 'Sizing' icon is visible, stating: "Control size-related settings such as element size, number of divisions along an edge, use of sphere or body of influence, minimum size, etc. Press F1 for help." The 'Details of "Sizing" - Sizing' panel is open at the bottom left, showing the following settings:

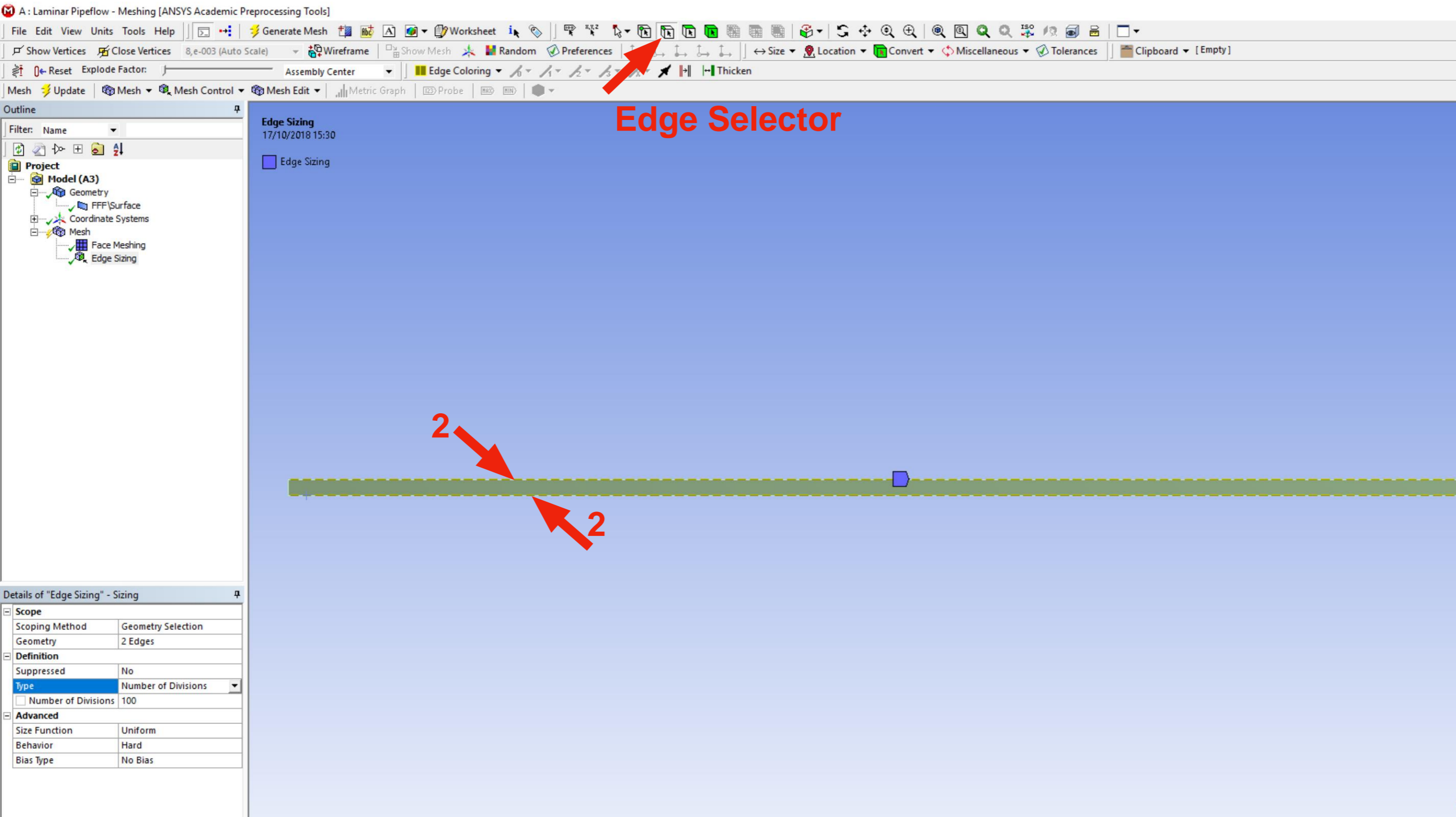
Details of "Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	Apply Cancel
Definition	
Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	Default (0,40003 m)
Advanced	
<input type="checkbox"/> Defeature Size	Default (2,0002e-003 m)
Behavior	Soft
<input type="checkbox"/> Growth Rate	Default (1,2)
Capture Curvature	No
Capture Proximity	No

The bottom status bar shows: "Control size-related settings such as element size, number of divisions along an edge, use of sphere or body of influence, minimum size, etc." The bottom right corner displays: "Messages pane 1 Face Selected: Area = 0,8 m² Metric (m, kg, N, s, V, A) Degrees rad/s Celsius". A scale bar at the bottom indicates 0,000, 0,750, 1,500, 2,250, and 3,000(m).

2- Add Sizing

Click on Edge selector/Select both bigger edges/Apply

Note! Use Ctrl key to add an edge/



Edge Sizing
17/10/2018 15:30

Edge Sizing

Edge Selector

2

2

Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges

Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	100

Advanced	
Size Function	Uniform
Behavior	Hard
Bias Type	No Bias

Sizing: Define Number of Divisions

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

Set Behavior to **Hard**

The screenshot displays the ANSYS Academic Preprocessing Tools interface. The main window shows a 3D model of a pipe with a mesh. The 'Edge Sizing' tool is active, and the 'Details of "Edge Sizing" - Sizing' panel is open. The panel shows the following configuration:

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges

Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	100

Advanced	
Size Function	Uniform
Behavior	Hard
Bias Type	No Bias

Red arrows point to the 'Type' dropdown menu, the 'Number of Divisions' input field, and the 'Behavior' dropdown menu.

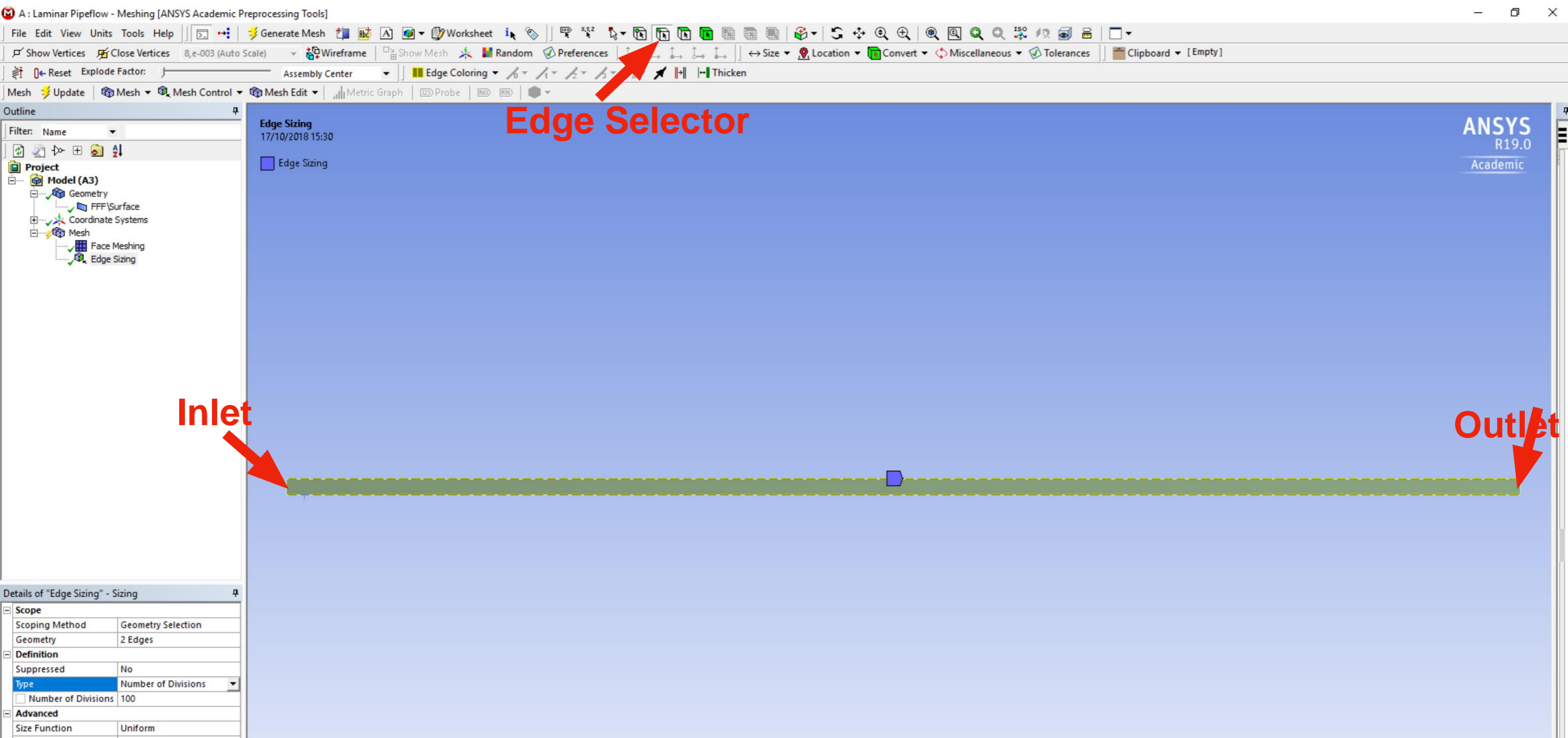
Add Sizing to Inlet and Outlet

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**



Edge Sizing

17/10/2018 15:30

Edge Sizing

Edge Selector

Inlet

Outlet

ANSYS R19.0 Academic

Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges

Definition

Suppressed	No
Type	Number of Divisions
Number of Divisions	100

Advanced

Size Function	Uniform
---------------	---------

Generate Mesh

Generate
Generate the mesh if out of date.
If only some mesh is out of date,
only that mesh will be generated.
Press F1 for help.

Outline

- Project*
- Model (A3)
 - Geometry
 - FFF\Surface
 - Materials
 - Coordinate Systems
 - Mesh
 - Face Meshing
 - Edge Sizing
 - Edge Sizing 2

Details of "Edge Sizing 2" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	5
Advanced	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

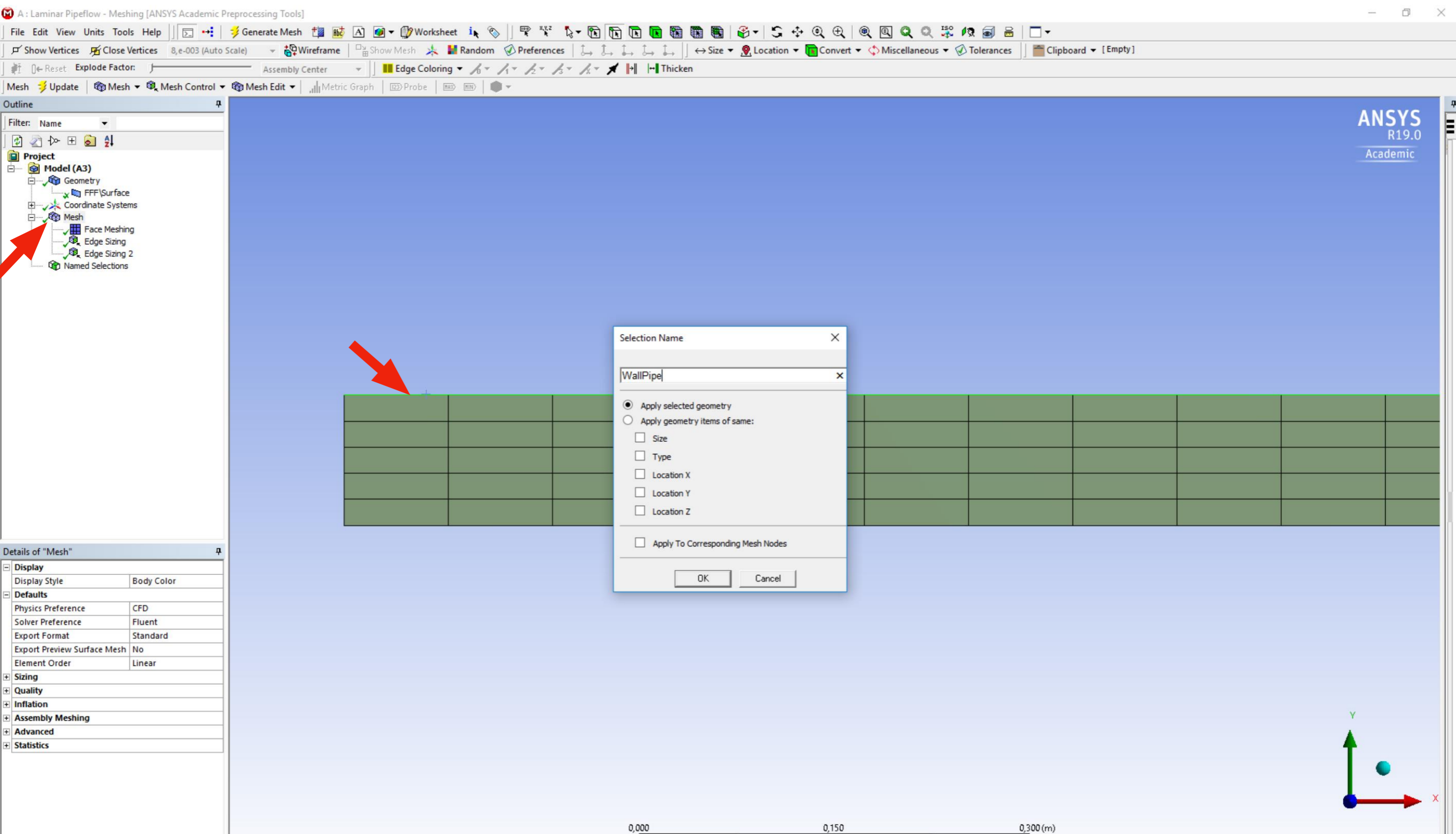
0,000 0,750 1,500 2,250 3,000(m)

Messages pane No Selection Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

Check the green marks

Create Named Selection

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



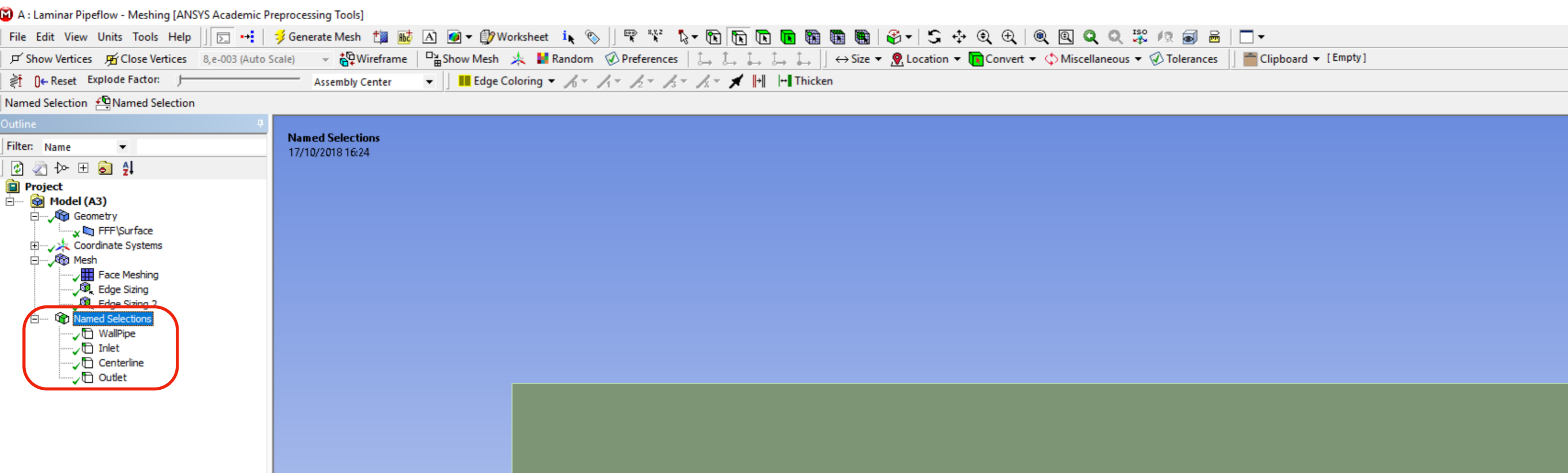
Create Named Selection for other edges

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)

The screenshot displays the ANSYS Workbench interface. On the left is the 'Toolbox' containing various analysis and component systems. The 'Project Schematic' area shows a hierarchical tree with components: Fluid Flow (Fluent), Geometry, Mesh, Setup, Solution, and Results. A right-click context menu is open over the 'Mesh' component, with the 'Update' option highlighted. Red arrows point to the 'Mesh' component and the 'Update' option. The text 'R-Click' is written in red above the context menu.

R-Click

Messages

	A	B	C	D
1	Type	Text	Association	Date/Time

Right-click to update component.

Job Monitor... Show Progress Hide 0 Messages

3- Fluent Launcher

Click on the Double Precision icon

The screenshot displays the ANSYS Workbench interface. On the left, the 'Toolbox' contains various analysis systems, with 'Fluid Flow (Fluent)' selected. The 'Project Schematic' shows a sequence of steps: 1. Fluid Flow (Fluent), 2. Geometry, 3. Mesh, 4. Setup, 5. Solution, and 6. Results. The 'Setup' step is highlighted with a dashed box, and a 'Double Precision' icon is visible next to it.

The 'Fluent Launcher 2021 R1 (Setting Edit Only)' dialog box is open, showing the following settings:

- Dimension:** 2D (selected), 3D
- Options:** Double Precision (checked), Display Mesh After Reading (checked), Do not show this panel again (unchecked), Load ACT (unchecked)
- Parallel (Local Machine):** Solver Processes: 1, Solver GPGPUs per Machine: 0

Below the settings, there are sections for 'What's New', 'Latest News', and 'Learning Resources'. The 'Start' button is highlighted with a tooltip that reads: 'Launch ANSYS Fluent with the selected options.'

At the bottom of the screen, the status bar shows 'Starting FLUENT' and several utility icons: Job Monitor..., No DPS Connection, Show Progress, and Show 0 Messages.

Check Mesh

Click on check/perform mesh check
Check the given information

The screenshot displays the ANSYS Academic Research Mechanical and CFD software interface. The title bar indicates the file path: A:\Laminar Pipeflow Parallel Fluent@wirepc9.intranet.epfl.ch [axi, dp, pbns, lam] [ANSYS Academic Research Mechanical and CFD]. The main menu bar includes File, Setting Up Domain, Setting Up Physics, User Defined, Solving, Postprocessing, Viewing, Parallel, and Design. The Mesh toolbar is visible, with a red arrow pointing to the 'Check' button. The Task Page shows the 'General' tab with the 'Mesh' section containing 'Scale...', 'Check', and 'Report Quality' buttons. The 'Solver' section is set to 'Pressure-Based' and 'Absolute' velocity formulation. The 'Time' section is set to 'Steady' and 'Axisymmetric'. The '2D Space' section is set to 'Axisymmetric Swirl'. The 'Gravity' checkbox is unchecked. The 'Units...' button is also visible. The main window shows a mesh of a pipe with a green line representing the mesh. The ANSYS R19.0 Academic logo is in the top right corner. The Console window at the bottom shows the following output:

```
Console
x-coordinate: min (m) = 0.000000e+00, max (m) = 8.000000e+00
y-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e-01
Volume statistics:
  minimum volume (m3): 1.600000e-03
  maximum volume (m3): 1.600000e-03
  total volume (m3): 8.000000e-01
Face area statistics:
  minimum face area (m2): 2.000000e-02
  maximum face area (m2): 8.000000e-02
Checking mesh.....
Done.
```

Setup/General

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

Click on **Axisymmetric** Checkbox

The screenshot displays the ANSYS Fluent software interface. The title bar shows the file path and solver settings. The main menu bar includes File, Domain, Physics, User-Defined, Solution, Results, View, Parallel, and Design. The toolbar contains various meshing and simulation tools. The Outline View on the left shows the tree structure with 'Setup' expanded to 'General'. The Task Page on the right shows the 'General' task page with the 'Mesh' section expanded. The 'Solver' section is visible, showing 'Type' set to 'Pressure-Based' and 'Velocity Formulation' set to 'Absolute'. The 'Time' section is expanded, showing 'Steady' selected and '2D Space' selected, with 'Axisymmetric' checked. A red arrow points to the 'Axisymmetric' checkbox. The main window shows a meshed geometry with a blue arrow indicating flow direction. The ANSYS 2021 R1 logo is visible in the bottom right corner.

File Domain Physics User-Defined Solution Results View Parallel Design

Mesh Zones Interfaces Mesh Models Turbo Model Adapt Surface

Display... Info Units... Check Quality Make Polyhedra Combine Delete... Append Mesh... Dynamic Mesh... Mixing Planes... Enable Turbo Topology... Turbo Create... Refine / Coarsen... Create Manage...

Outline View

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
- Cell Registers
- Initialization
- Calculation Activities
- Run Calculation

Results

- Surfaces
- Graphics
- Plots
- Animations
- Reports

Parameters & Customization

Simulation Reports

Task Page

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

ANSYS 2021 R1

0 selected all

Setup/Models

At the Tree Menu double click on **Models**
Double Click on Viscous - Choose Laminar
Select Laminar and OK

The screenshot displays the ANSYS Fluent software interface. The title bar indicates the file path: A:\Laminar Pipeflow Parallel Fluent@wirepc9.intranet.epfl.ch [axi, dp, pbns, lam] [ANSYS Academic Research Mechanical and CFD]. The main menu bar includes File, Setting Up Domain, Setting Up Physics, User Defined, Solving, Postprocessing, Viewing, Parallel, and Design. The toolbar is divided into sections: Mesh (Display..., Info, Units..., Check, Repair, Quality, Improve..., Scale..., Transform, Make Polyhedra), Zones (Combine, Separate, Adjacency..., Delete..., Deactivate..., Activate..., Append, Replace Mesh..., Replace Zone...), Interfaces (Mesh..., Overset...), Mesh Models (Dynamic Mesh..., Mixing Planes..., Turbo Topology...), Adapt (Mark/Adapt Cells, Manage Registers..., More), and Surface (Create, Manage...). The Tree panel on the left shows a hierarchy: Setup > Models. The Models list includes Multiphase - Off, Energy - Off, Viscous (Laminar), Radiation - Off, Heat Exchanger - Off, Species - Off, Discrete Phase - Off, Solidification & Melting - Off, Acoustics - Off, and Electric Potential - Off. The Viscous (Laminar) item is selected. The Task Page panel shows the Models list with Viscous - Laminar selected. A dialog box titled 'Viscous Model' is open, showing the 'Model' section with 'Laminar' selected. Other options include Inviscid, Spalart-Allmaras (1 eqn), k-epsilon (2 eqn), k-omega (2 eqn), Transition k-k-omega (3 eqn), Transition SST (4 eqn), Reynolds Stress (5 eqn), Scale-Adaptive Simulation (SAS), and Detached Eddy Simulation (DES). The OK, Cancel, and Help buttons are visible at the bottom of the dialog box. The main window shows a blue mesh visualization of a pipe flow with a green horizontal line representing the flow direction.

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

The screenshot displays the ANSYS Fluent software interface. The top menu bar includes File, Setting Up Domain, Setting Up Physics, User Defined, Solving, Postprocessing, Viewing, Parallel, and Design. The toolbar contains various meshing and material-related tools. The Tree panel on the left shows the project hierarchy, with 'Materials' selected under 'Setup'. The Task Page on the right shows the 'Materials' list, with 'air' selected. A 'Create/Edit Materials' dialog box is open, showing the following settings:

- Name: air
- Material Type: fluid
- Chemical Formula: air
- Mixture: none
- Order Materials by: Name (selected)
- Properties:
 - Density (kg/m3): constant, value: 1
 - Viscosity (kg/m-s): constant, value: 0.002

Buttons at the bottom of the dialog include Change/Create, Delete, Close, and Help.

Setup/Boundary Conditions

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

Left Click on **Centerline**

Change Type to **Axis** then OK

The screenshot shows the software interface for setting up boundary conditions. The 'Boundary Conditions' panel is open, and the 'centerline' boundary is selected. The 'Type' dropdown menu is open, showing 'axis' as the selected option. The 'Phase' is set to 'mixture' and the 'ID' is 7. The main window shows a blue domain with a green centerline. The console displays simulation statistics.

Phase	Type	ID
mixture	wall	7
	axis	
	exhaust-fan	
	inlet-vent	
	intake-fan	
	interface	
	mass-flow-inlet	
	mass-flow-outlet	
	outflow	
	outlet-vent	
	overset	

```
x-coordinate: min (m) = 0.000000e+00, max (m) = 8.000000e+00
y-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e-01
Volume statistics:
  minimum volume (m3): 1.600000e-03
  maximum volume (m3): 1.600000e-03
  total volume (m3): 8.000000e-01
Face area statistics:
  minimum face area (m2): 2.000000e-02
  maximum face area (m2): 8.000000e-02
Checking mesh.....
```

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

The screenshot displays the ANSYS Fluent software interface. The top menu bar includes File, Setting Up Domain, Setting Up Physics, User Defined, Solving, Postprocessing, Viewing, Parallel, and Design. The main toolbar is divided into sections: Mesh (Display..., Info, Check, Quality, Transform, Units..., Repair, Improve..., Make Polyhedra), Zones (Combine, Separate, Adjacency..., Delete..., Deactivate..., Activate..., Append, Replace Mesh..., Replace Zone...), Interfaces (Mesh..., Overset...), Mesh Models (Dynamic Mesh..., Mixing Planes..., Turbo Topology...), Adapt (Mark/Adapt Cells, Manage Registers..., More), and Surface (+ Create, Manage...). The Tree panel on the left shows the hierarchy: Setup > Boundary Conditions > inlet (velocity-inlet, id=6). The Task Page shows the Boundary Conditions list with 'inlet' selected. A 'Velocity Inlet' dialog box is open, showing the 'Momentum' tab. The 'Zone Name' is 'inlet'. The 'Velocity Specification Method' is set to 'Components', and the 'Reference Frame' is 'Absolute'. The 'Axial-Velocity (m/s)' is set to '1' with a 'constant' profile, and the 'Radial-Velocity (m/s)' is set to '0' with a 'constant' profile. The 'Supersonic/Initial Gauge Pressure (pascal)' is set to '0' with a 'constant' profile. The 'OK', 'Cancel', and 'Help' buttons are visible at the bottom of the dialog box.

Setup/Boundary Conditions

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

Left Click on **Outlet**

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

The screenshot displays the ANSYS Fluent software interface. On the left, the 'Tree' panel shows the 'Boundary Conditions' folder expanded, with 'outlet (pressure-outlet, id=8)' selected. The 'Task Page' panel shows the 'Boundary Conditions' task page with 'outlet' selected in the 'Zone' list. The 'Type' dropdown menu is set to 'pressure-outlet', and the 'ID' is 8. A red arrow points to the 'pressure-outlet' dropdown. The 'Console' panel at the bottom right shows the following statistics:

```
Console
x-coordinate: min (m) = 0.000000e+00, max (m) = 8.000000e+00
y-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e-01
Volume statistics:
  minimum volume (m3): 1.600000e-03
  maximum volume (m3): 1.600000e-03
  total volume (m3): 8.000000e-01
Face area statistics:
  minimum face area (m2): 2.000000e-02
```

Setup/Boundary Conditions

At the Tree Menu double click on **Boundary Conditions**
Left Click on **WallPipe** - Check if the Type is **Wall**

The screenshot displays the ANSYS Fluent software interface. The title bar shows the file path: A:\Laminar Pipeflow Parallel Fluent@wirepc9.intranet.epfl.ch [axi, dp, pbns, lam] [ANSYS Academic Research Mechanical and CFD]. The main menu bar includes File, Setting Up Domain, Setting Up Physics, User Defined, Solving, Postprocessing, Viewing, Parallel, and Design. The toolbar contains various icons for meshing and simulation. The Tree Menu on the left shows the hierarchy: Setup > Boundary Conditions > wallpipe (wall, id=5). The Boundary Conditions panel in the center lists several zones: centerline, inlet, interior-fff_surface, outlet, and wallpipe. The wallpipe zone is selected, and its properties are shown at the bottom: Phase: mixture, Type: wall, ID: 5. A red arrow points to the Type dropdown menu. The main window shows a mesh of a pipe with a green line representing the centerline. A scale bar at the bottom right indicates 0 to 2 meters.

Solution/Methods

At the Tree Menu double click on Solution/Methods
Change the Scheme to **SIMPLE**
Check if the Momentum is **Second Order Upwind**

The screenshot displays the ANSYS Fluent software interface. The title bar indicates the file path: "B:\Copy of Laminar Pipeflow Parallel Fluent@wirepc9.intranet.epfl.ch [axi, dp, pbns, lam] [ANSYS Academic Research Mechanical and CFD]". The main menu bar includes "File", "Setting Up Domain", "Setting Up Physics", "User Defined", "Solving", "Postprocessing", "Viewing", "Parallel", and "Design".

The "Task Page" panel is open, showing the "Solution Methods" section. The "Scheme" dropdown is set to "SIMPLE". Under "Spatial Discretization", the "Momentum" dropdown is set to "Second Order Upwind". Other options include "Pressure-Velocity Coupling", "Transient Formulation", and checkboxes for "Non-Iterative Time Advancement", "Frozen Flux Formulation", "Pseudo Transient", "Warped-Face Gradient Correction", and "High Order Term Relaxation".

The "Tree" panel on the left shows the project hierarchy, with "Solution" > "Methods" selected. The "Mesh" panel on the right shows a blue mesh of a pipe with a red horizontal line indicating a boundary. A scale bar at the bottom right indicates a length of 2 (m).

Solution/Monitors (Residuals Control)

At the Tree Menu double click on **Monitors**
Double Click on **Residuals**
Change all the Criteria to **1e-6**

A:Laminar Pipeflow Parallel Fluent@wirepc9.intranet.epfl.ch [axi, dp, pbns, lam] [ANSYS Academic Research Mechanical and CFD]

File Setting Up Domain Setting Up Physics User Defined Solving Postprocessing Viewing Parallel Design

Mesh Zones Interfaces Mesh Models Adapt Surface

Display... Info Check Quality Transform Scale... Separate Deactivate... Append Replace Mesh... Mesh... Overset... Dynamic Mesh... Mixing Planes... Mark/Adapt Cells Manage Registers... Create Manage... Units... Repair Improve... Make Polyhedra Adjacency... Activate... Replace Zone... Turbo Topology... More

Tree

- Setup
 - General
 - Models
 - Materials
 - Fluid
 - air
 - Solid
 - Cell Zone Conditions
 - Boundary Conditions
 - centerline (axis, id=7)
 - inlet (velocity-inlet, id=6)
 - interior-fff_surface (interior, id=1)
 - outlet (pressure-outlet, id=8)
 - wallpipe (wall, id=5)
 - Dynamic Mesh
 - Reference Values
- Solution
 - Methods
 - Controls
 - Report Definitions
 - Monitors
 - Residual
 - Report Files
 - Report Plots
 - Convergence Conditions
 - Cell Registers
 - Initialization
 - Calculation Activities
 - Run Calculation
- Results
 - Graphics
 - Plots
 - Animations
 - Reports
 - Parameters & Customization

Task Page

Monitors

Report Definition quantities can be monitored during solution when they are included in Report Files or Report Plots.

Specifying Convergence Conditions allows you to define stop conditions for the solver based on Report Definition convergence.

Mesh

Residual Monitors

Equations	Residual	Monitor Check	Convergence Absolute Criteria
continuity		<input checked="" type="checkbox"/>	1e-6
x-velocity		<input checked="" type="checkbox"/>	1e-6
y-velocity		<input checked="" type="checkbox"/>	1e-6

Options

- Print to Console
- Plot

Window

1 Curves... Axes...

Iterations to Plot

1000

Iterations to Store

1000

Residual Values

- Normalize
- Scale
- Compute Local Scale

Iterations

5

Convergence Criterion

absolute

Convergence Conditions...

OK Plot Renormalize Cancel Help

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

The screenshot displays the ANSYS Fluent software interface. The title bar shows the file path: A:\Laminar Pipeflow Parallel Fluent@wirepc9.intranet.epfl.ch [axi, dp, pbns, lam] [ANSYS Academic Research Mechanical and CFD]. The main menu bar includes File, Setting Up Domain, Setting Up Physics, User Defined, Solving, Postprocessing, Viewing, Parallel, and Design. Below the menu bar are several toolbars: Mesh (Display..., Info, Check, Quality, Transform, Scale..., Units..., Repair, Improve..., Make Polyhedra), Zones (Combine, Delete..., Append, Separate, Deactivate..., Replace Mesh..., Adjacency..., Activate..., Replace Zone...), Interfaces (Mesh..., Overset...), Mesh Models (Dynamic Mesh..., Mixing Planes..., Turbo Topology...), Adapt (Mark/Adapt Cells, Manage Registers..., More), and Surface (Create, Manage...). The Tree panel on the left shows a hierarchical view of the model setup, with 'Initialization' selected under the 'Solution' folder. The Task Page panel on the right is titled 'Solution Initialization' and contains the following settings: 'Initialization Methods' with 'Standard Initialization' selected; 'Compute from' set to 'inlet'; 'Reference Frame' set to 'Relative to Cell Zone'; and 'Initial Values' for Gauge Pressure (pascal) set to 0, Axial Velocity (m/s) set to 1, and Radial Velocity (m/s) set to 0. At the bottom of the Task Page are buttons for 'Initialize', 'Reset', 'Patch...', 'Reset DPM Sources', and 'Reset Statistics'. The main window shows a blue mesh of a pipe with a green horizontal line representing the inlet.

Solution/Run Calculation

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

Set the Number of Iterations to **100**

Click on the **Calculate** icon

The screenshot displays the ANSYS Fluent software interface. The title bar shows the file path: "A:\Fluid Flow (Fluent) Parallel Fluent@wirepc9.intranet.epfl.ch [axi, dp, pbns, lam, single-process] [CFD Solver - Level 2 - CFD Solver - Level 1]". The main menu bar includes File, Domain, Physics, User-Defined, Solution, Results, View, Parallel, and Design. The toolbar contains various icons for meshing and simulation. The Outline View on the left shows a tree structure with categories like Discrete Phase, Materials, Cell Zone Conditions, Boundary Conditions, Mesh Interfaces, Reference Values, Reference Frames, Named Expressions, and Solution. The Solution category is expanded, showing Methods, Controls, Report Definitions, and Monitors. The Run Calculation dialog box is open, showing the following settings:

- Run Calculation**: Check Case..., Update Dynamic Mesh...
- Pseudo Transient Settings**:
 - Fluid Time Scale**: Time Step Method (Automatic), Time Scale Factor (1), Length Scale Method (Conservative), Verbosity (0).
- Parameters**: Number of Iterations (100), Reporting Interval (1), Profile Update Interval (1).
- Solution Processing**: Statistics (Data Sampling for Steady Statistics), Data File Quantities...
- Solution Advancement**: Calculate

Two red arrows point to the "Number of Iterations" field and the "Calculate" button. A green arrow points to the "Calculate" button in the toolbar.

3- Control residuals

A: Laminar Pipeflow Parallel Fluent@wirepc9.intranet.epfl.ch [axi, dp, pbns, lam] [ANSYS Academic Research Mechanical and CFD]

File Setting Up Domain Setting Up Physics User Defined Solving Postprocessing Viewing Parallel Design

Mesh Zones Interfaces Mesh Models Adapt Surface

Display... Check Quality Scale... Transform... Combine Delete... Append... Mesh... Dynamic Mesh... Mark/Adapt Cells Create...
Info Repair Improve... Make Polyhedra Separate Deactivate... Replace Mesh... Overset... Mixing Planes... Manage Registers... Manage...
Units... Repair Improve... Make Polyhedra Adjacency... Activate... Replace Zone... Turbo Topology... More

Tree Filter Text

- Setup
 - General
 - Models
 - Materials
 - Fluid
 - air
 - Solid
 - Cell Zone Conditions
 - Boundary Conditions
 - centerline (axis, id=7)
 - inlet (velocity-inlet, id=6)
 - interior-fff_surface (interior, id=1)
 - outlet (pressure-outlet, id=8)
 - wallpipe (wall, id=5)
 - Dynamic Mesh
 - Reference Values
- Solution
 - Methods
 - Controls
 - Report Definitions
 - Monitors
 - Cell Registers
 - Initialization
 - Calculation Activities
 - Run Calculation
- Results
 - Graphics
 - Plots
 - Scene
 - Animations
 - Reports
 - Parameters & Customization

Task Page

Run Calculation

Check Case... Update Dynamic Mesh...

Number of Iterations: 100 Reporting Interval: 1

Profile Update Interval: 1

Data File Quantities... Acoustic Signals...

Calculate Help

Residuals

- continuity
- x-velocity
- y-velocity

Iterations

ANSYS R19.0 Academic

Console

```
43 8.5582e-06 2.3036e-08 2.5029e-09 0:00:00 57
44 6.4454e-06 1.6532e-08 1.9213e-09 0:00:00 56

iter continuity x-velocity y-velocity time/iter
45 4.9324e-06 1.1849e-08 1.4380e-09 0:00:00 55
46 3.7792e-06 8.4902e-09 1.0571e-09 0:00:00 54
47 2.8704e-06 6.0746e-09 7.9405e-10 0:00:00 53
48 2.1358e-06 4.3605e-09 6.0101e-10 0:00:00 52
49 1.5939e-06 3.1358e-09 4.5901e-10 0:00:00 51
50 1.1951e-06 2.2463e-09 3.5431e-10 0:00:00 50
! 51 solution is converged
51 8.9671e-07 1.6012e-09 2.6639e-10 0:00:00 49
Writing data to C:\Users\kardan\AppData\Local\Temp\WB_WIREPC9_kardan_23668_2\unsaved_project_files\dp0\FFF\Fluent\FFF.ip ...
x-coord
y-coord
pressure
x-velocity
y-velocity
Done.
Calculation complete.
```

Run Results from Workbench

The screenshot displays the ANSYS Workbench software interface. The main window is titled "Project Schematic" and shows a project named "A". The project schematic is a tree view with the following items:

- 1 Fluid Flow (Fluent)
- 2 Geometry
- 3 Mesh
- 4 Setup
- 5 Solution
- 6 Results
- 7 Parameters

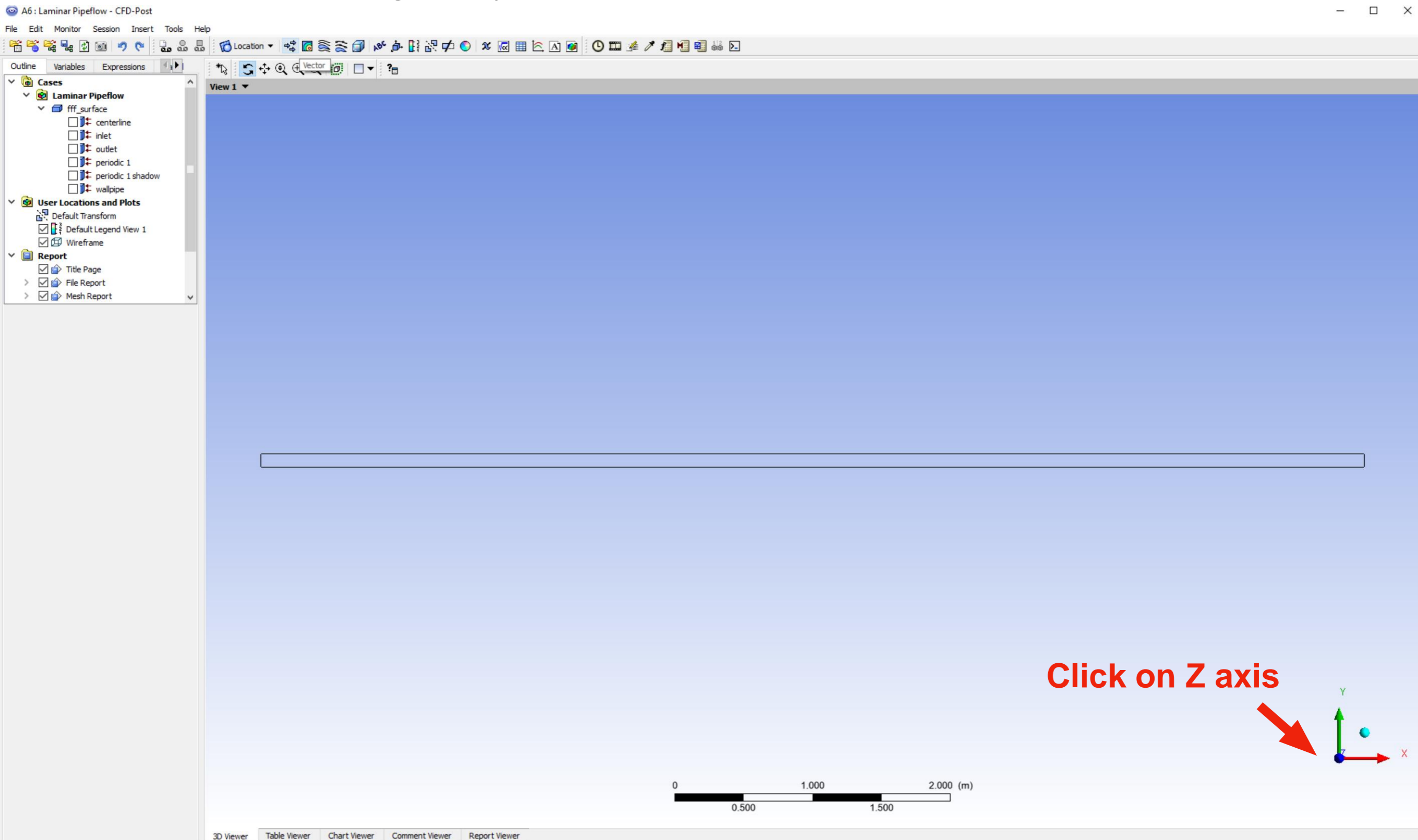
Below the schematic, the text "Laminar Pipeflow" is visible. A red arrow points from the "Results" item to a "Parameter Set" box. The "Toolbox" on the left lists various analysis systems, with "Fluid Flow (Fluent)" selected. The "Messages" panel at the bottom shows a table with the following data:

	A	B	C	D
1	Type	Text	Association	Date/Time

At the bottom of the interface, there is a status bar with the text "Starting CFD-Post..." and buttons for "Job Monitor...", "Show Progress", and "Hide 0 Messages".

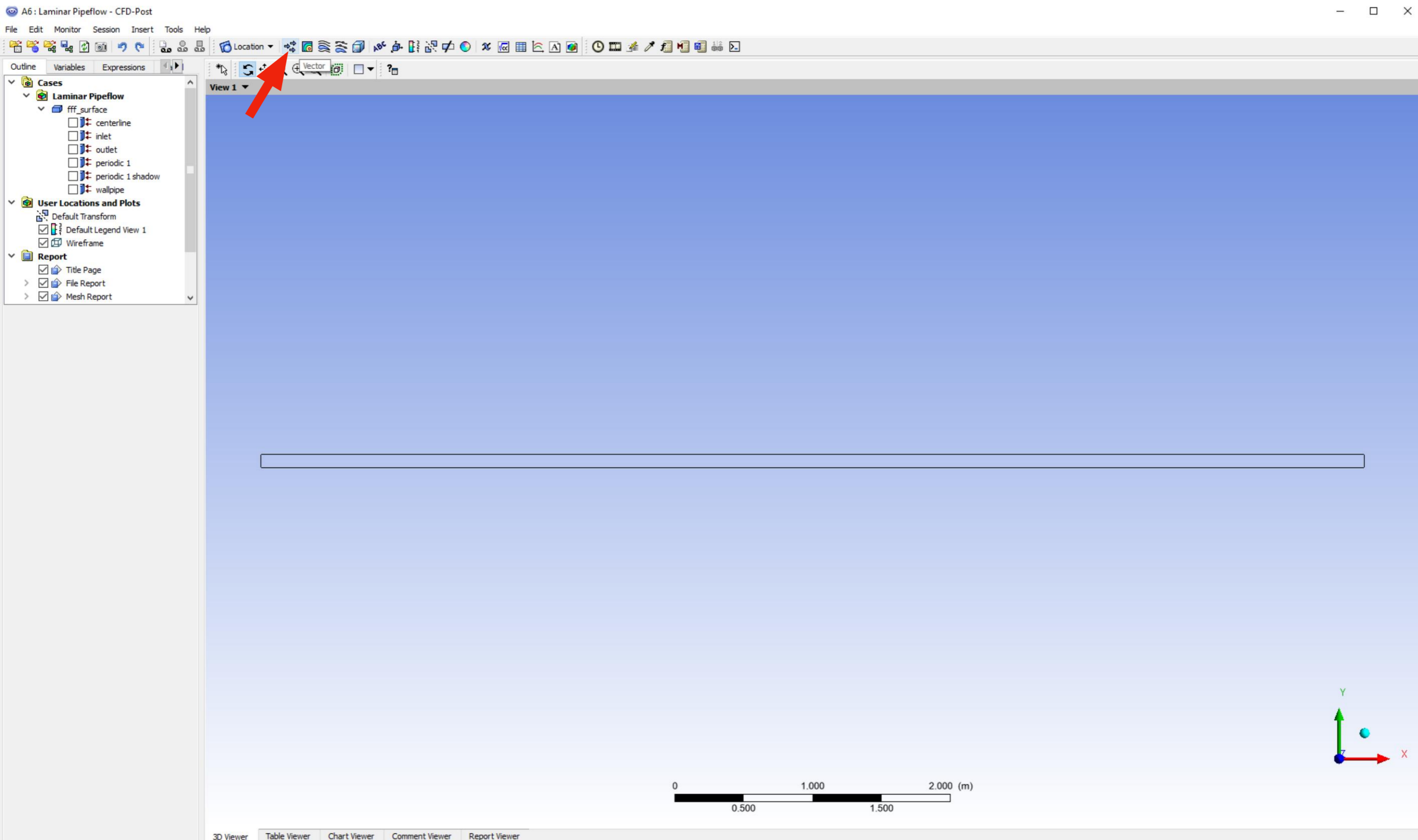
4- CFD-Post (Objective: Velocity Vectors)

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

The screenshot displays the ANSYS CFD-Post interface for a laminar pipe flow simulation. The main window shows a 3D visualization of velocity vectors in a pipe. A color scale on the left indicates velocity magnitude in m s^{-1} , ranging from $0.000\text{e}+000$ (blue) to $1.924\text{e}+000$ (red). The vectors are plotted in a horizontal pipe, showing a parabolic velocity profile. The 'Details of Vector 1' panel is highlighted with a red circle and an arrow pointing to the 'Locations' dropdown menu, which is set to 'periodic 1'. The 'Variable' is set to 'Velocity' and the 'Boundary Data' is set to 'Conservative'. The 'Projection' is set to 'None'. A scale bar at the bottom indicates a length of 0.300 m.

Velocity Vector 1

1.924e+000

1.443e+000

9.619e-001

4.809e-001

0.000e+000

[m s⁻¹]

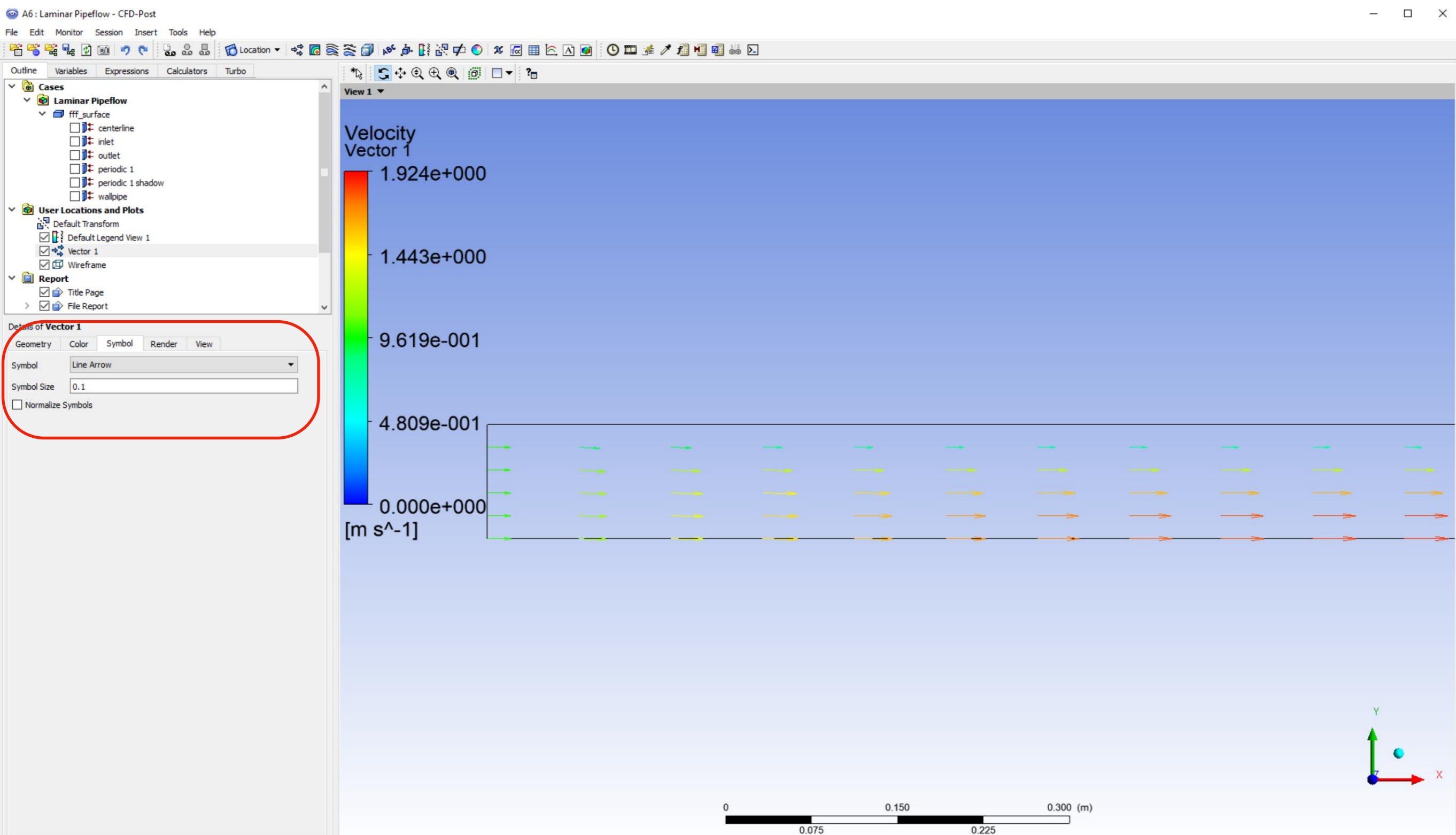
0 0.075 0.150 0.225 0.300 (m)

Apply Reset Defaults

CFD-Post (Objective: Velocity Vectors)

At the Details Menu click on **Symbol**

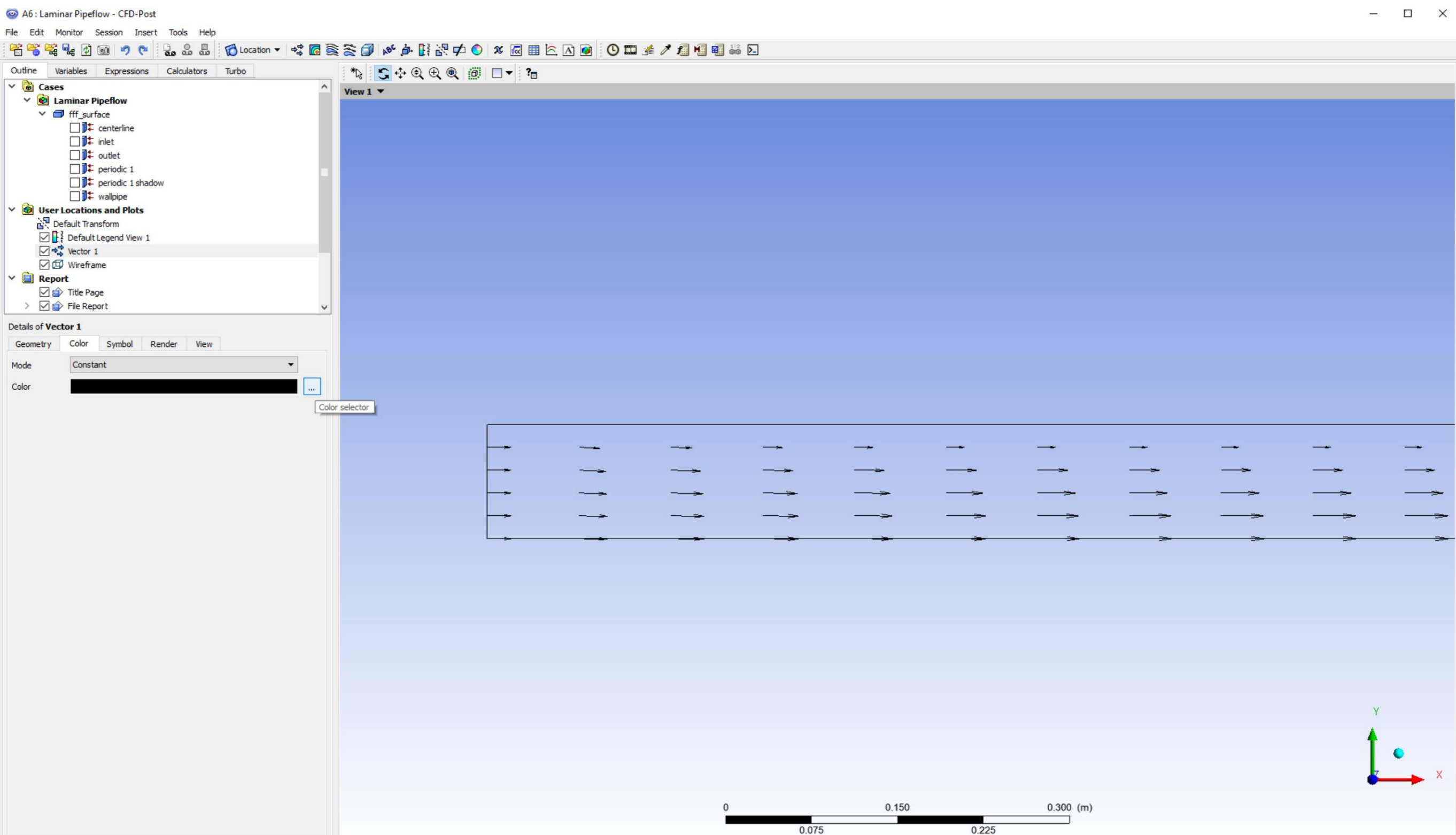
Change the symbol size to **0.1** and Apply



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

The screenshot displays the ANSYS CFD-Post interface for a laminar pipe flow simulation. The main window shows a 3D view of the pipe with a velocity contour plot. A red arrow points to the 'Contour' icon in the toolbar. The 'Details of Contour 1' panel is visible, showing the following settings:

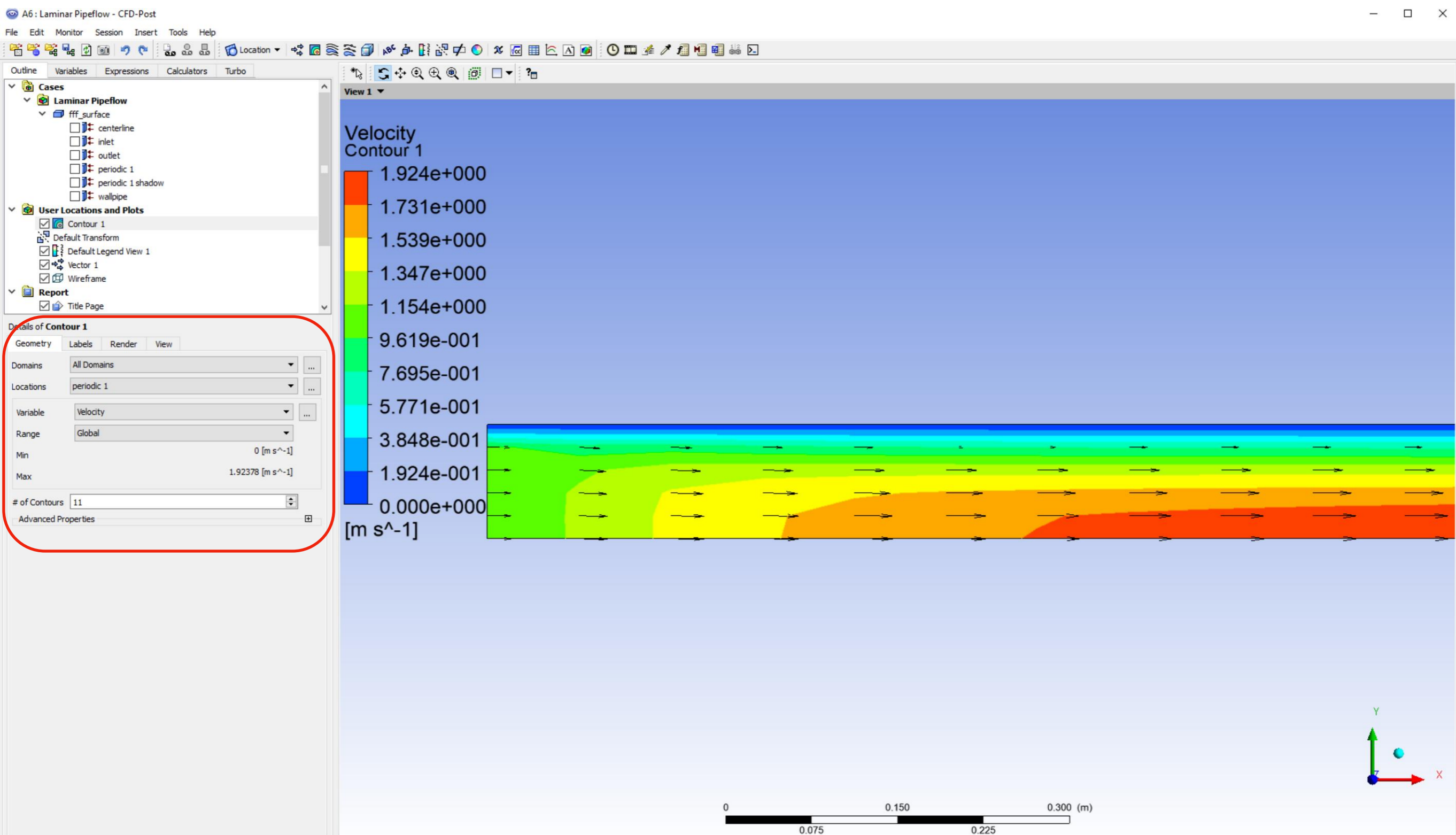
- Geometry: All Domains
- Locations: centerline
- Variable: Pressure
- Range: Global
- Min: unknown
- Max: unknown
- # of Contours: 11

The main view shows a 3D model of a pipe with a velocity contour plot. A scale bar at the bottom indicates 0 to 0.300 meters. The plot shows a velocity profile across the pipe, with a maximum velocity of approximately 0.300 m/s. The velocity is zero at the walls and increases towards the centerline.

CFD-Post (Objective: Velocity Contour)

Set the Location to **Periodic 1**

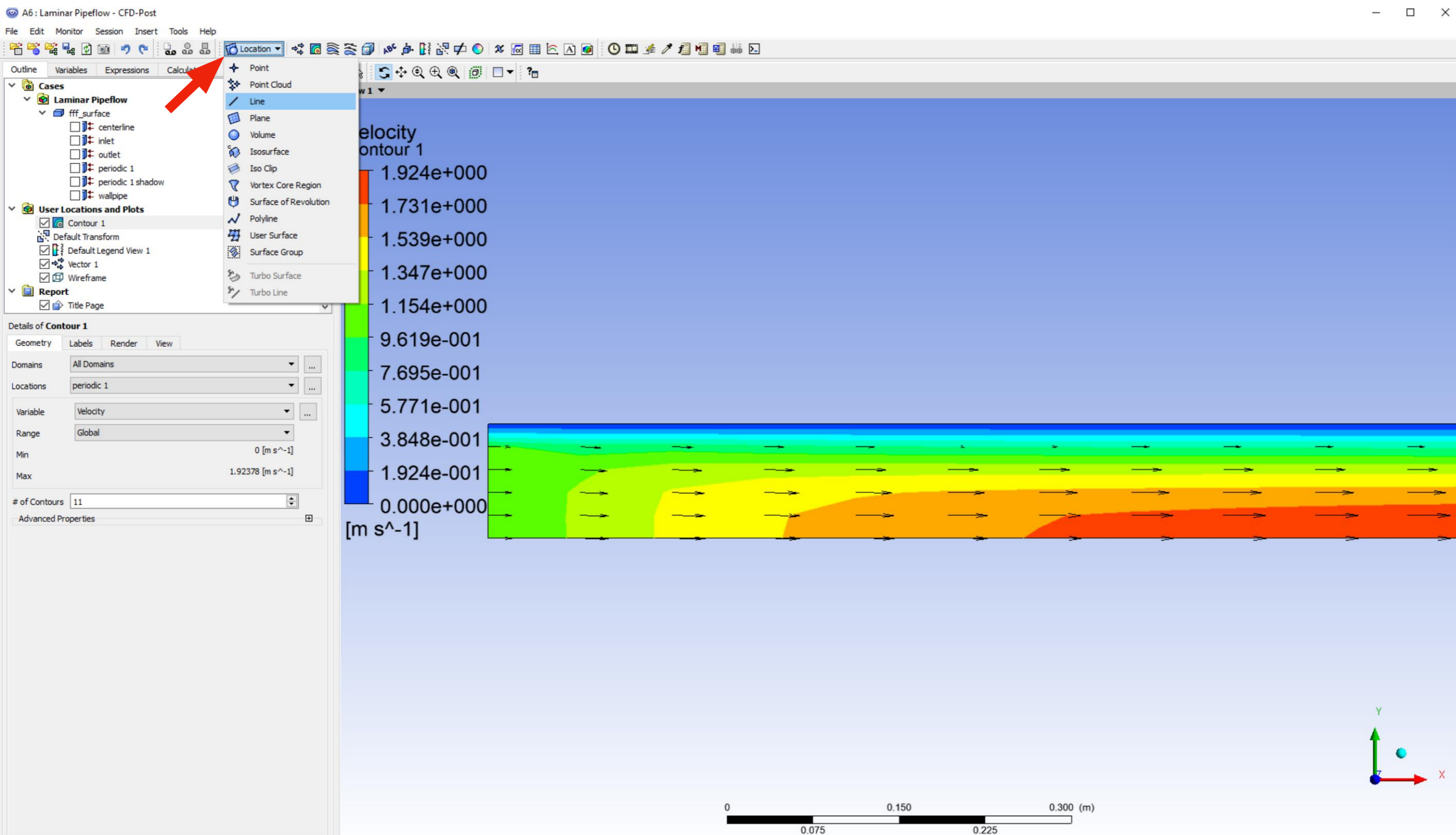
Change the Variable to **Velocity**, then Apply



CFD-Post (Objective: Creating Chart)

Location/Line

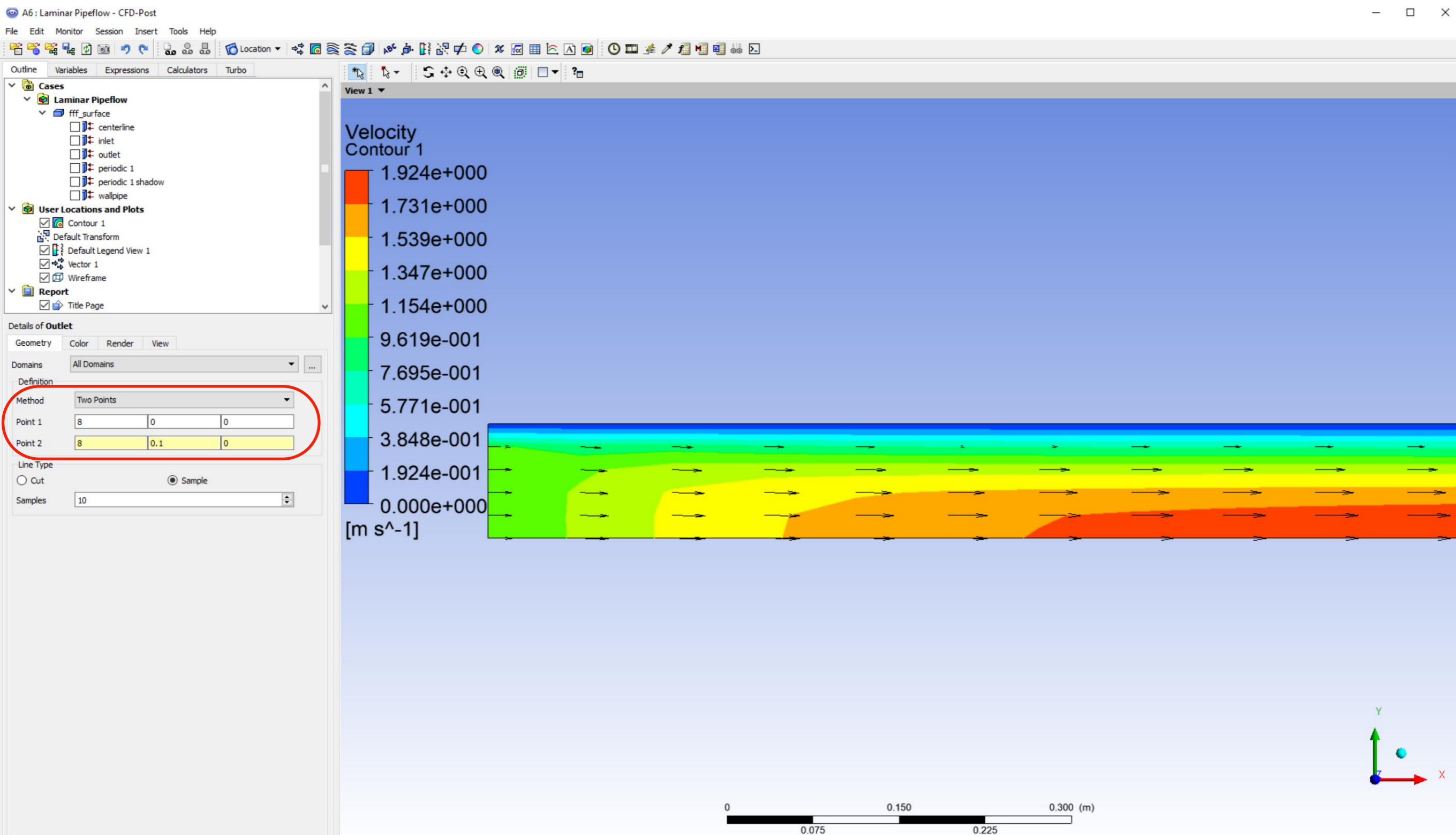
Change the Name to **Outlet**, then Ok



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

The screenshot displays the CFD-Post software interface. On the left, the 'Outline' panel shows a tree view with 'Cases' expanded to 'Laminar Pipeflow', then 'fff_surface', and 'User Locations and Plots'. Under 'User Locations and Plots', 'Outlet' is checked. Below the outline is the 'Details of Chart 1' window, which has tabs for 'General', 'Data Series', 'X Axis', 'Y Axis', 'Line Display', and 'Chart Display'. The 'Data Series' tab is active, showing 'Series 1 (Outlet)' in a list. Below this, the 'Name' field contains 'Series 1'. The 'Data Source' section has 'Location' selected, and the dropdown menu shows 'Outlet'. At the bottom of the window are 'Apply', 'Export', 'Reset', and 'Defaults' buttons. On the right, a large chart grid is visible with a red arrow pointing to the top toolbar. The grid has a vertical axis from 0 to 1,000 and a horizontal axis from 0 to 1,000. At the bottom of the chart area, there are tabs for '3D Viewer', 'Table Viewer', 'Chart Viewer', 'Comment Viewer', and 'Report Viewer'.

CFD-Post (Objective: Creating Chart)

At the Details Menu click on **Data Series**
Change the Location to **Outlet**, then Apply

The screenshot displays the ANSYS CFD-Post software interface. On the left, the 'Outline' pane shows a project structure with 'Cases' > 'Laminar Pipeflow' > 'fff_surface' containing various locations like 'centerline', 'inlet', 'outlet', 'periodic 1', 'periodic 1 shadow', and 'wallpipe'. Below this, 'User Locations and Plots' includes 'Contour 1', 'Default Transform', 'Default Legend View 1', 'Outlet', 'Vector 1', and 'Wireframe'. The 'Report' pane is also visible.

The main area shows a 'Details of Chart 1' panel with tabs for 'General', 'Data Series', 'X Axis', 'Y Axis', 'Line Display', and 'Chart Display'. The 'Data Series' tab is active, showing 'Series 1 (Outlet)'. A red arrow points to the 'Outlet' option in the 'Data Source' dropdown menu. Other options include 'Location', 'File', and 'Monitor Data'. The 'Name' field is set to 'Series 1'. At the bottom of the panel are 'Apply', 'Export', 'Reset', and 'Defaults' buttons.

The background features a large empty grid for plotting, with axes ranging from 0 to 1,000 on both the X and Y dimensions.

CFD-Post (Objective: Creating Chart)

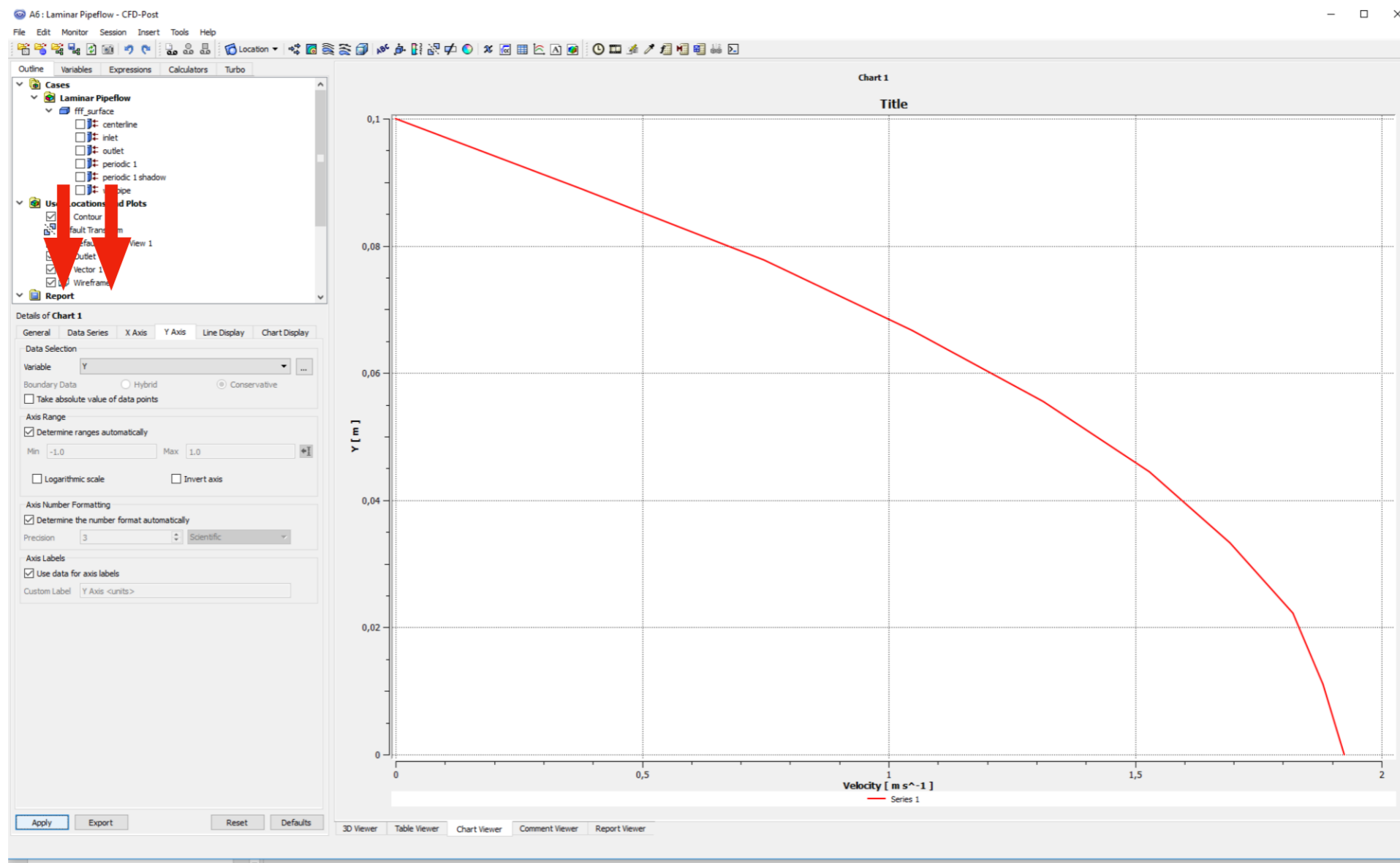
At the Details Menu click on **X Axis**

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

The screenshot displays the ANSYS Workbench interface for a CFD simulation. The main window shows a mesh of a pipe. The 'Reference Values' dialog box is open, allowing the user to set reference parameters. A red arrow points to the 'Density (kg/m3)' field, which is set to 1. The 'Velocity (m/s)' field is also set to 1. The 'Reference Values' section includes fields for Area (m2), Density (kg/m3), Enthalpy (j/kg), Length (m), Pressure (pascal), Temperature (k), Velocity (m/s), Viscosity (kg/m-s), and Ratio of Specific Heats. The 'Reference Zone' dropdown is currently empty. The 'Tree' view on the left shows the 'Reference Values' node selected under the 'Solution' folder. The 'Console' at the bottom shows the following text:

```
domains,  
mixture  
zones,  
outlet  
inlet
```

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

The screenshot displays the ANSYS Fluent software interface. The main window shows a mesh of a pipe with a green line representing the wall shear stress. The 'Run Calculation' dialog box is open, showing the following settings:

- Number of Iterations: 100
- Reporting Interval: 1
- Profile Update Interval: 1
- Buttons: Check Case..., Update Dynamic Mesh..., Calculate, Help

A warning dialog box is overlaid on the 'Run Calculation' dialog, with a red arrow pointing to the 'Use settings changes for current and future calculations' option. The warning dialog text reads:

Settings have changed!
Warning: The settings have changed in Fluent. How would you like to proceed?
 Use settings changes for current calculation only.
 Use settings changes for current and future calculations.
Buttons: OK, Cancel, Help

The console window at the bottom shows the following text:

```
domains,  
mixture,  
zones,  
outlet  
inlet  
wallpipe  
interior-fff_surface  
centerline  
fff_surface
```

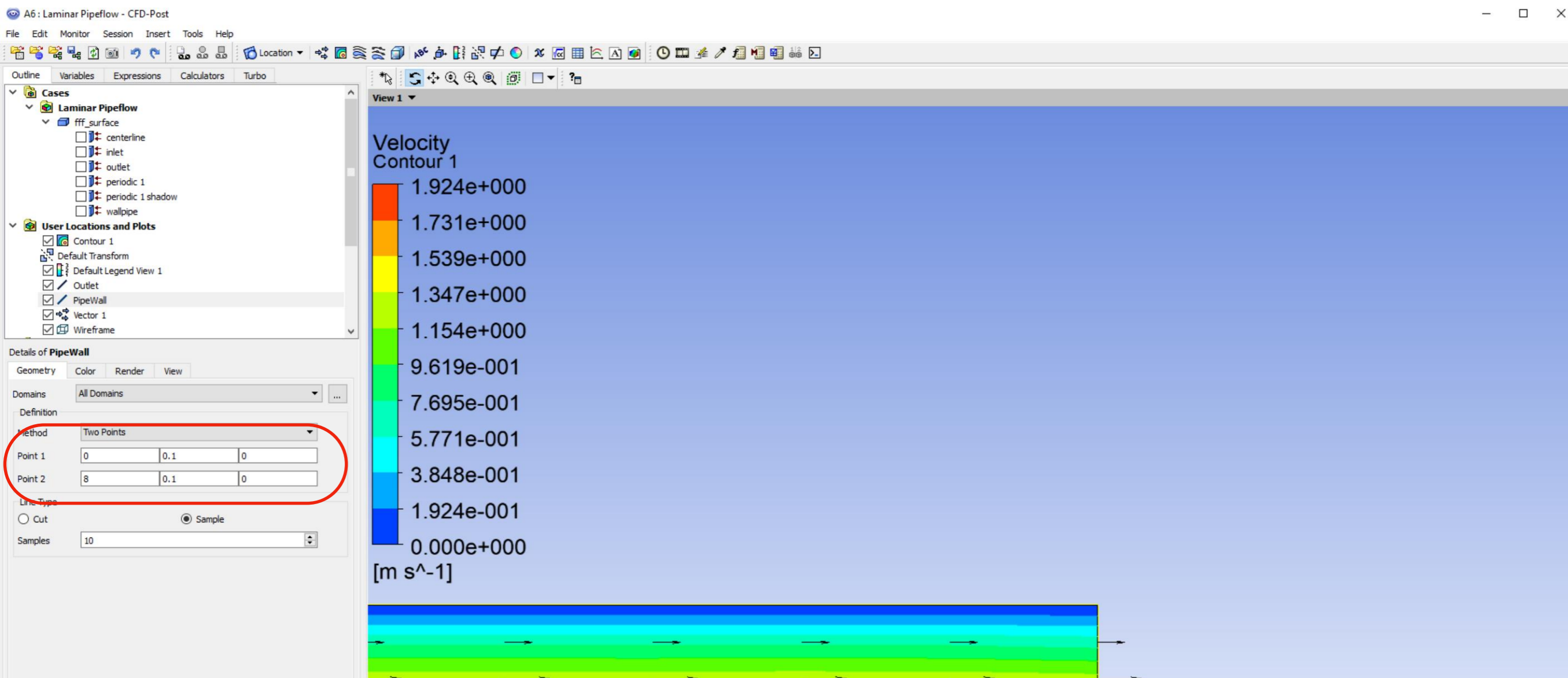
CFD-Post (Objective: Wall Shear)

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



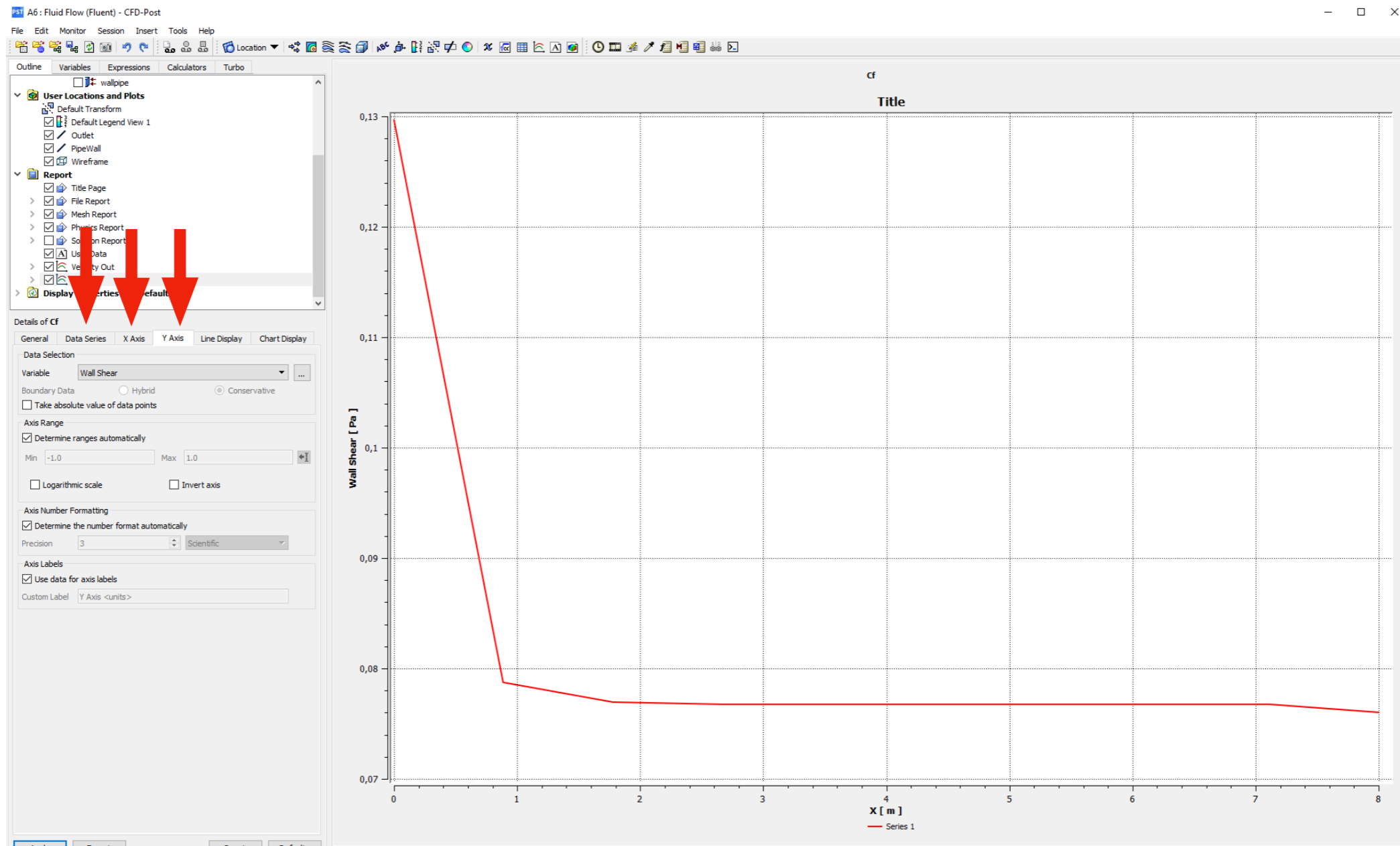
CFD-Post (Objective: Wall Shear)

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

The screenshot displays the ANSYS Fluent software interface. On the left, the 'Tree Outline' shows the 'Fluxes' option under the 'Reports' menu. The main window shows a 3D model of a pipe with a green centerline. A 'Flux Reports' dialog box is open, showing the 'Mass Flow Rate' option selected. The 'Boundaries' list includes 'centerline', 'inlet', 'interior-fff_surface', 'outlet', and 'wallpipe'. The 'Results' column shows values for each boundary. Below the dialog box, the console output shows the 'Mass Flow Rate (kg/s)' for the 'inlet', 'outlet', and 'wallpipe' boundaries, with the 'inlet' and 'outlet' values circled in red.

Boundary	Mass Flow Rate (kg/s)
centerline	0.03141592653589936
inlet	-0.03141592655417017
interior-fff_surface	-0
outlet	-0
wallpipe	-0

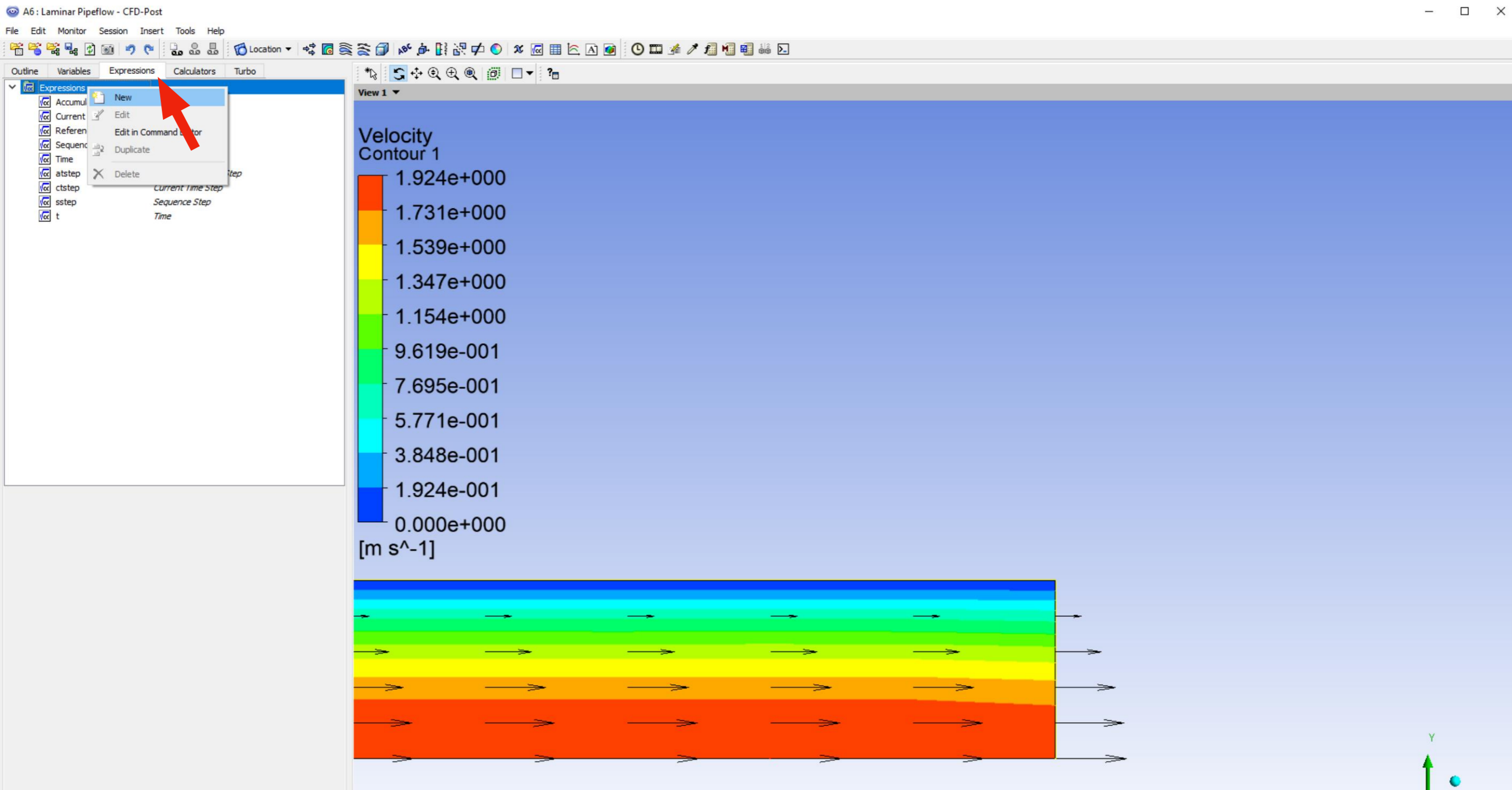
Boundary	Mass Flow Rate (kg/s)
inlet	0.031415927
outlet	-0.031415927
wallpipe	-0

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)

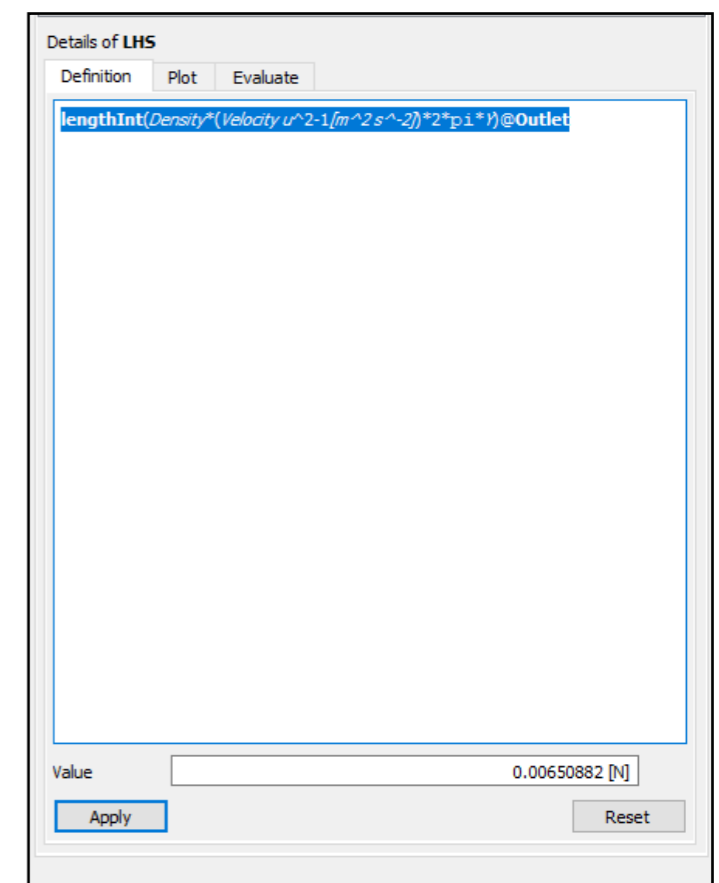
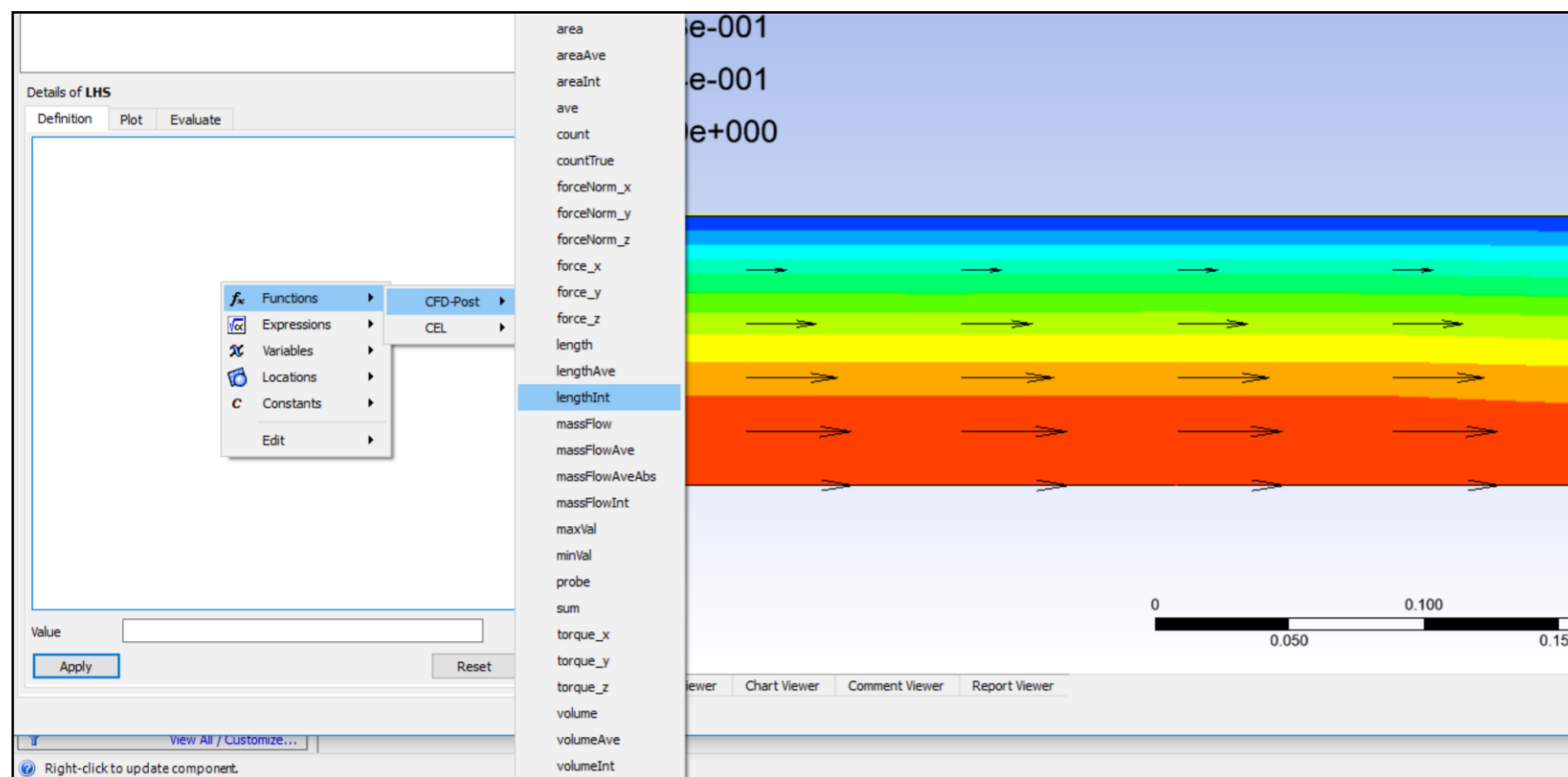
R-Click on Details of LHS and select Functions/CFD-Post/**LengthInt**

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

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**



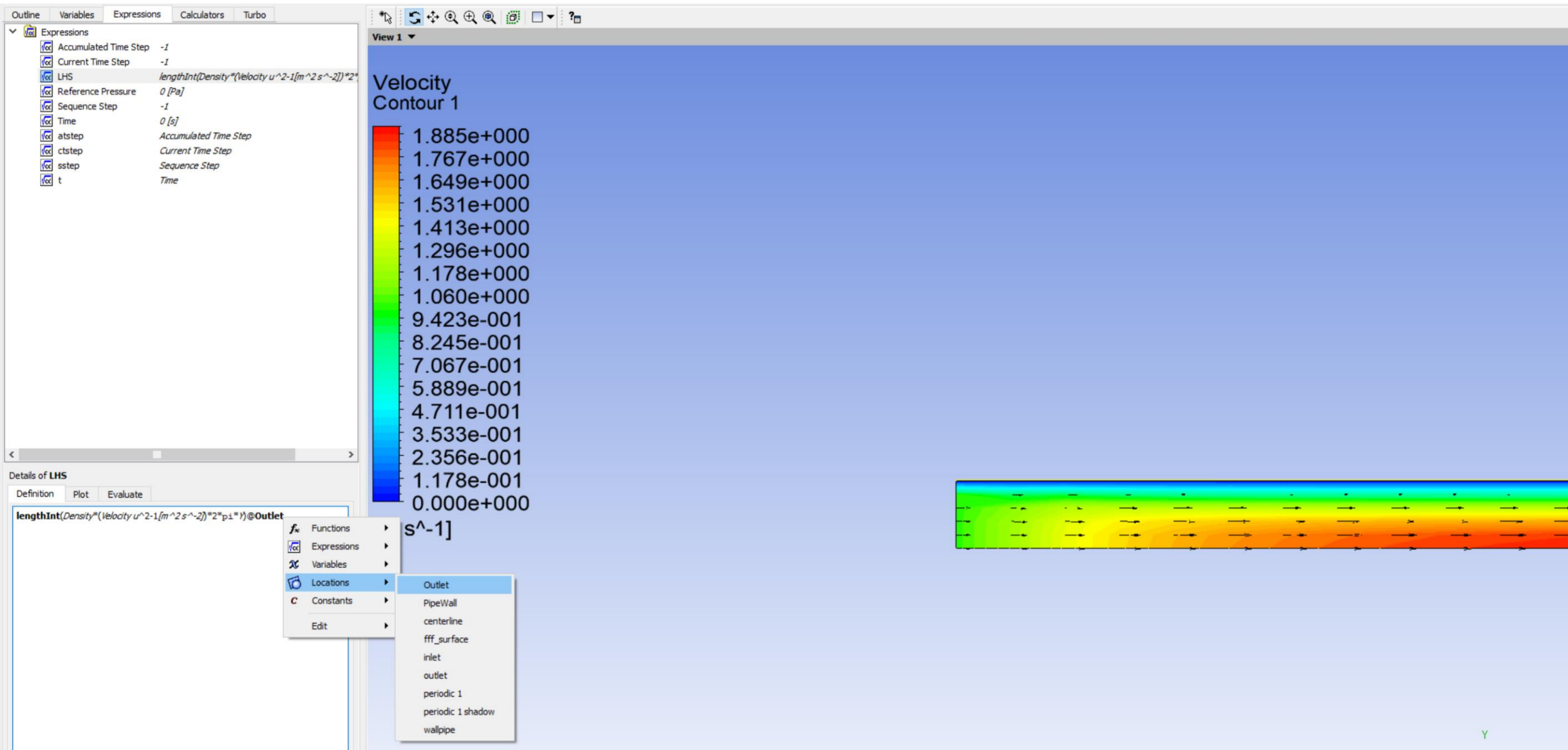
CFD-Post (Objective: Pressure Force)

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/**LengthInt**

Define:

$\text{lengthInt}(\text{Pressure} * 2 * \pi * Y) @ \text{Inlet} - \text{lengthInt}(\text{Pressure} * 2 * \pi * Y) @ \text{Outlet}$

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**

The screenshot displays the CFD-Post software interface. On the left, a 'Definition' panel is visible, containing a text input field with the expression: $\text{lengthInt}(\text{Pressure} * 2 * \pi * Y) @ \text{Inlet} - \text{lengthInt}(\text{Pressure} * 2 * \pi * Y) @ \text{Outlet}$. A red arrow points from the text 'RHS' below to this expression. Below the input field, a 'Value' field shows '0.403299 [N]' and buttons for 'Apply' and 'Reset'. The main area of the software is a 3D viewer showing a blue background with a coordinate system (X, Y, Z) and a scale bar from 0 to 2.000 (m). At the bottom, there are tabs for '3D Viewer', 'Table Viewer', 'Chart Viewer', 'Comment Viewer', and 'Report Viewer'.

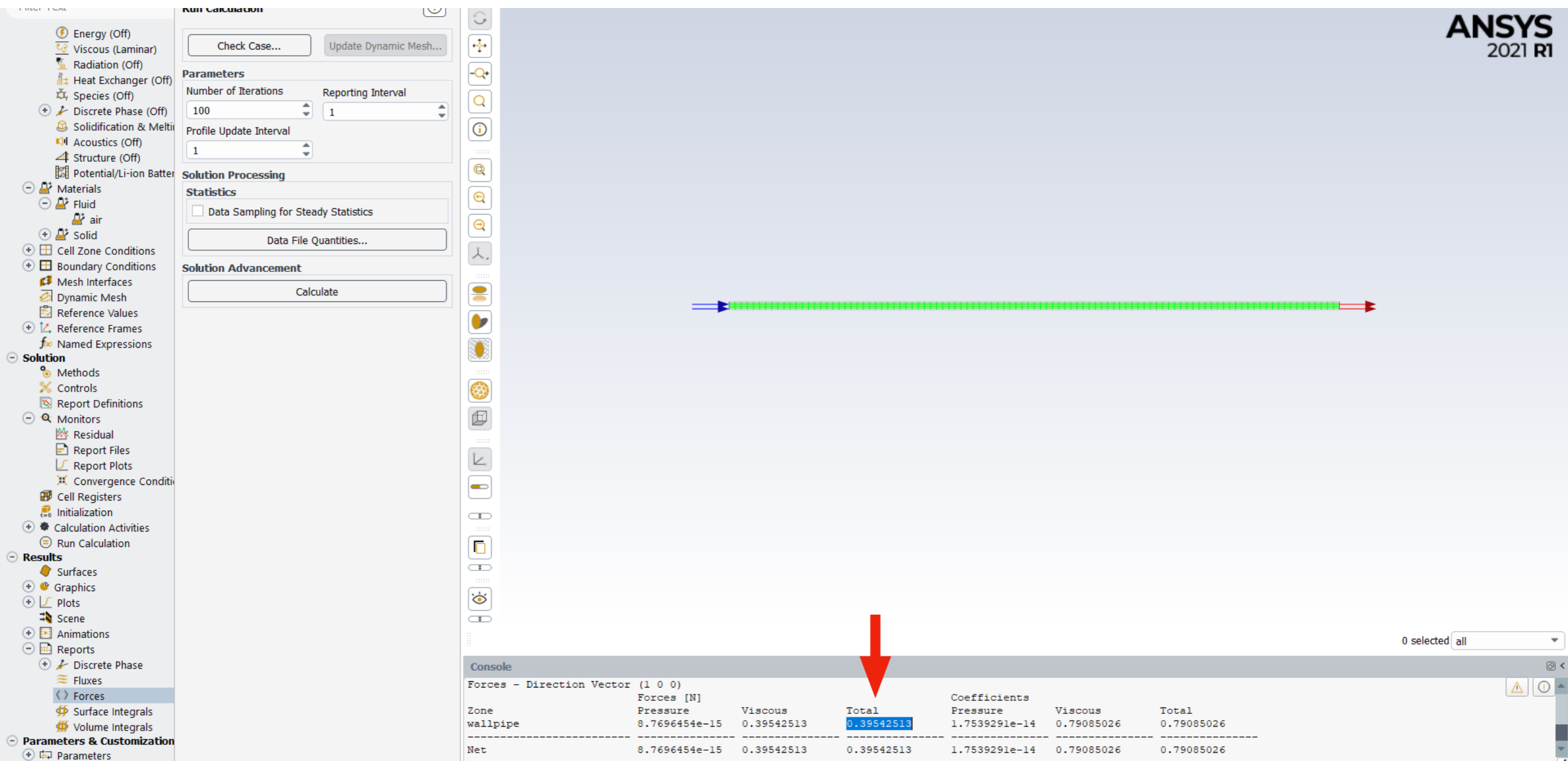
CFD-Post (Objective: Drag Force)

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



The screenshot displays the ANSYS Workbench interface. On the left, the Tree Menu is expanded to 'Results' > 'Reports' > 'Forces'. The 'Run Calculation' dialog is open, showing parameters for the simulation. The 'Console' window at the bottom displays the output for the 'Forces' report, with a red arrow pointing to the 'Total' force value for the 'wallpipe' zone.

Run Calculation Parameters:

- Number of Iterations: 100
- Reporting Interval: 1
- Profile Update Interval: 1

Console Output:

Forces - Direction Vector (1 0 0)						
Zone	Pressure Forces [N]	Viscous Forces [N]	Total Forces [N]	Pressure Coefficients	Viscous Coefficients	Total Coefficients
wallpipe	8.7696454e-15	0.39542513	0.39542513	1.7539291e-14	0.79085026	0.79085026
Net	8.7696454e-15	0.39542513	0.39542513	1.7539291e-14	0.79085026	0.79085026

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: $(\text{RHS}-\text{LHS}-F_{\text{shear}})/F_{\text{shear}}*100$

The screenshot displays the 'Details of MomBalance' dialog box in the CFD-Post software. The 'Definition' tab is active, showing the mathematical expression $(\text{RHS}-\text{LHS}-F_{\text{shear}})/F_{\text{shear}}*100$ in a text field. A red arrow points to this field. Below the field, the 'Value' is displayed as 0.345185. The 'Apply' button is highlighted with a blue border, and the 'Reset' button is visible. The background shows a 3D viewer with a blue gradient and a coordinate system (X, Y, Z) with a scale bar from 0 to 2.000 (m).

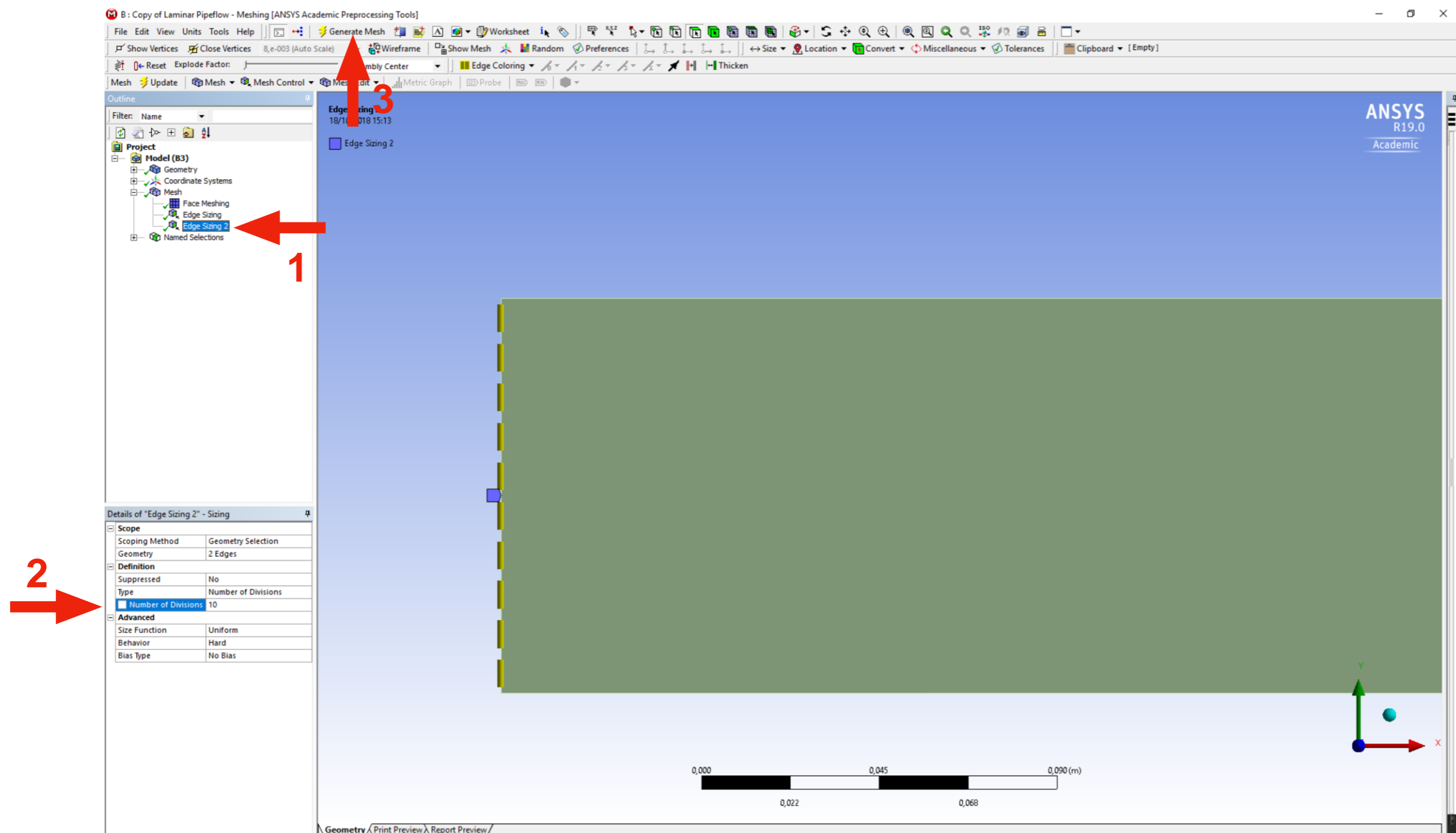
CFD-Post (Objective: Results Optimization - Mesh Resolution)

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**



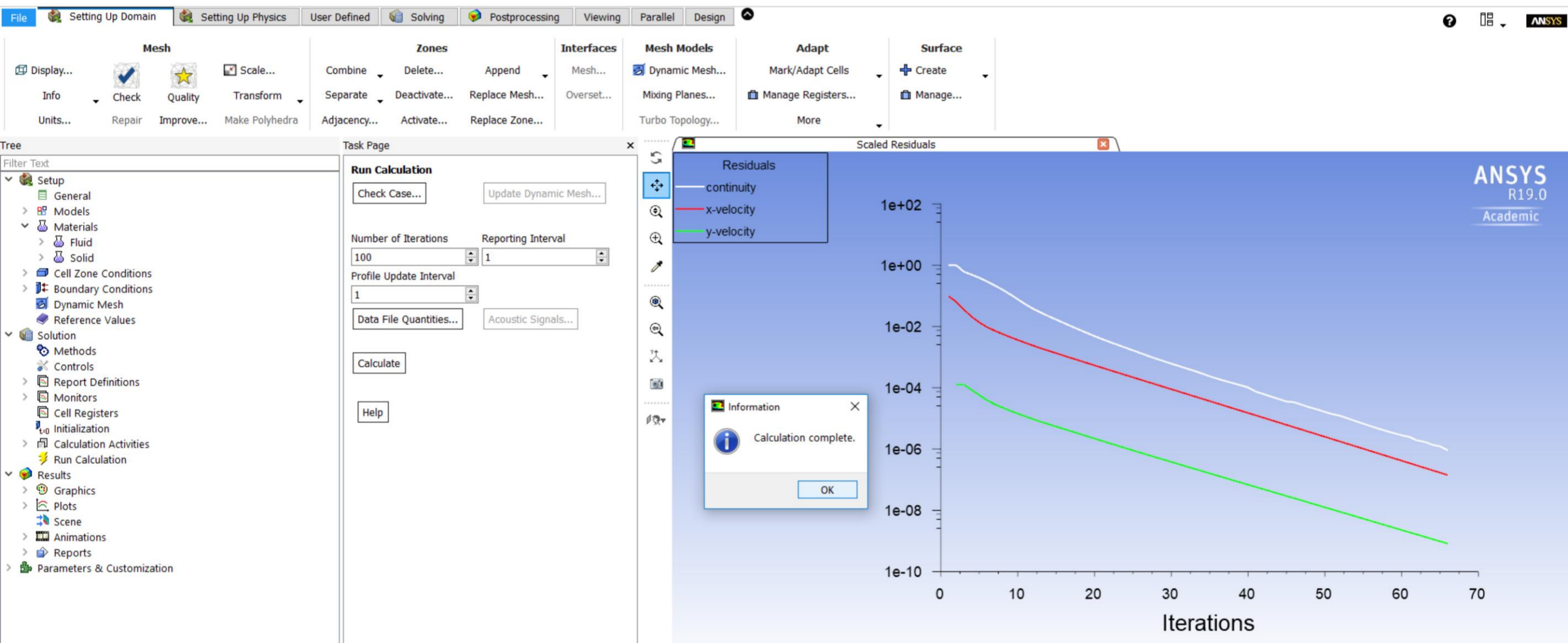
CFD-Post (Objective: Results Optimization)

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

The screenshot displays the ANSYS 2021 R1 interface. On the left is the Tree Menu, with 'Results' > 'Reports' > 'Forces' selected. The 'Run Calculation' panel is visible, showing parameters like 'Number of Iterations' (100) and 'Reporting Interval' (1). A 'Force Reports' dialog box is open, with 'Options' set to 'Forces', 'Direction Vector' (X: 1, Y: 0, Z: 0), and 'Wall Zones' set to 'wallpipe'. A red arrow points to the 'Print' button in this dialog. The console at the bottom shows the following output:

Forces - Direction Vector (1 0 0)						
	Forces [N]			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
wallpipe	9.0310911e-15	0.4084946	0.4084946	1.8062182e-14	0.8169892	0.8169892
Net	9.0310911e-15	0.4084946	0.4084946	1.8062182e-14	0.8169892	0.8169892

A red arrow points to the 'Total' value of 0.4084946 in the console output.

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

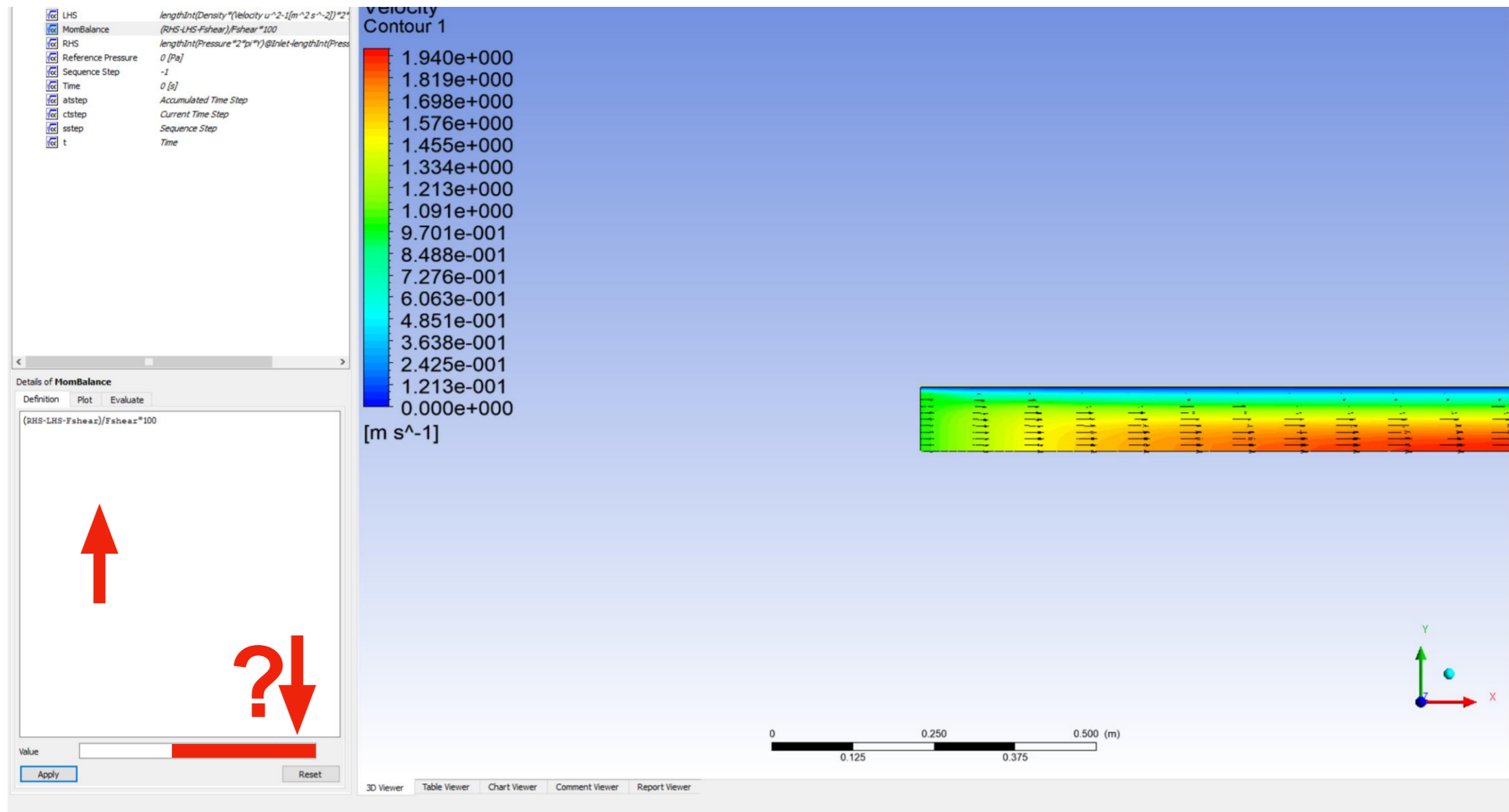
The screenshot displays the ANSYS CFD-Post interface. On the left, the 'Details of Fshear' panel is visible, showing the 'Definition' tab. The expression is set to 0.4084946 [N] , with a red arrow pointing to the value. Below the expression, the 'Value' field shows 0.408495 [N] and an 'Apply' button is present. The main 3D view shows a horizontal cylinder on a blue background. A coordinate system with X, Y, and Z axes is visible in the bottom right corner. A scale bar at the bottom indicates distances of 0, 0.750, 1.500, 2.250, and 3.000 meters. The bottom status bar includes options for 3D Viewer, Table Viewer, Chart Viewer, Comment Viewer, and Report Viewer.

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

