



Politechnika Wrocławska

Mechanical and Power Engineering Faculty
Full-time studies

Selected problems of thermal-flow processes

Exercise no. 4

Modeling of condensation

Wrocław 2020

TABLE OF CONTENTS

1. Introduction	2
2. Two-dimensional condensation on a flat plate.....	3
2.1. Geometry	3
2.2. Numerical Mesh.....	9
2.3. Numerical model.....	26
2.4. Calculations	48
2.5. Elaboration of the results	52
3. References.....	65

1. INTRODUCTION

The exercise will show how to model steam condensation on a flat horizontal plate. Modeling phase change phenomena (condensation, evaporation) is very difficult. In this exercise, one of the condensation models called *Wall Condensation Model* [1] will be presented. A mixture of saturated steam and air (non-condensing gas) flows around a flat plate at 90 °C at a speed of 0.1 m/s. The temperature of the mixture is 100 °C and the mass fraction of steam is 95%. In order to reduce the calculation time, the case will be modeled as two-dimensional. The diagram of the analyzed case is presented in Fig. 1.

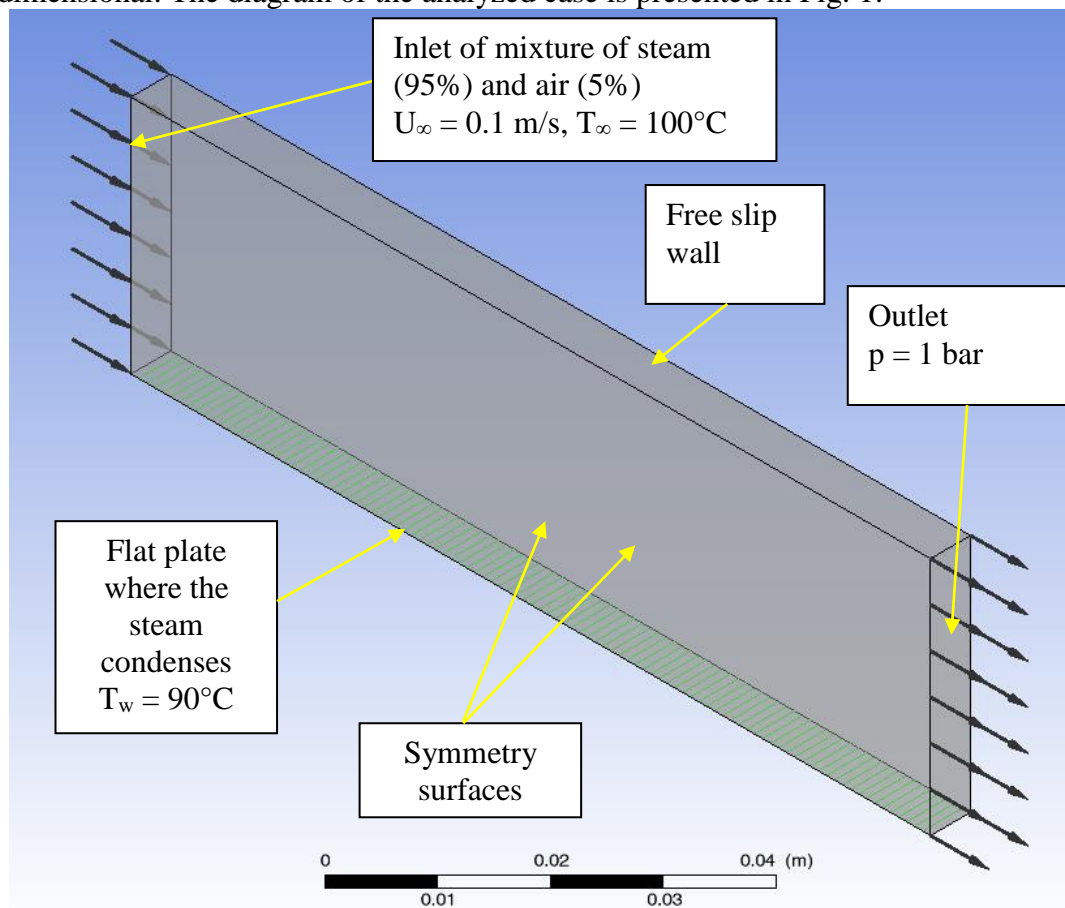


Fig. 1. Scheme of the issue of condensation on a flat plate

2. TWO-DIMENSIONAL CONDENSATION ON A FLAT PLATE

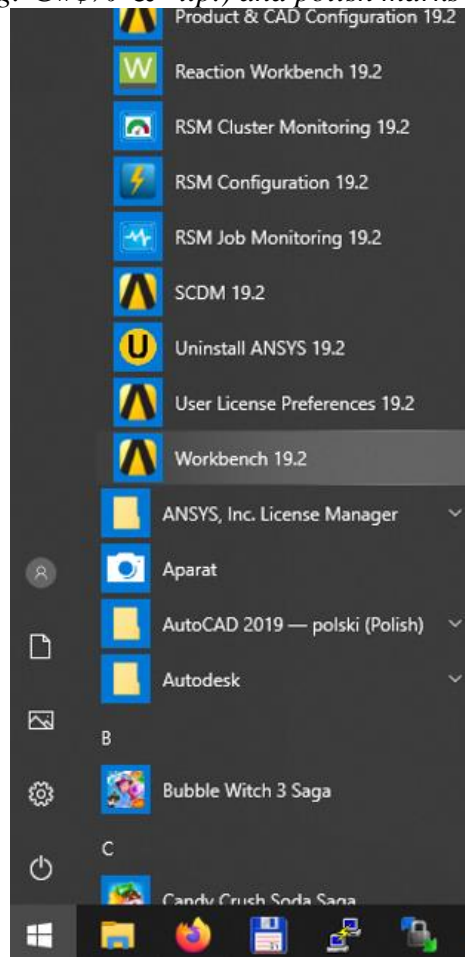
2.1. GEOMETRY

Do the following:

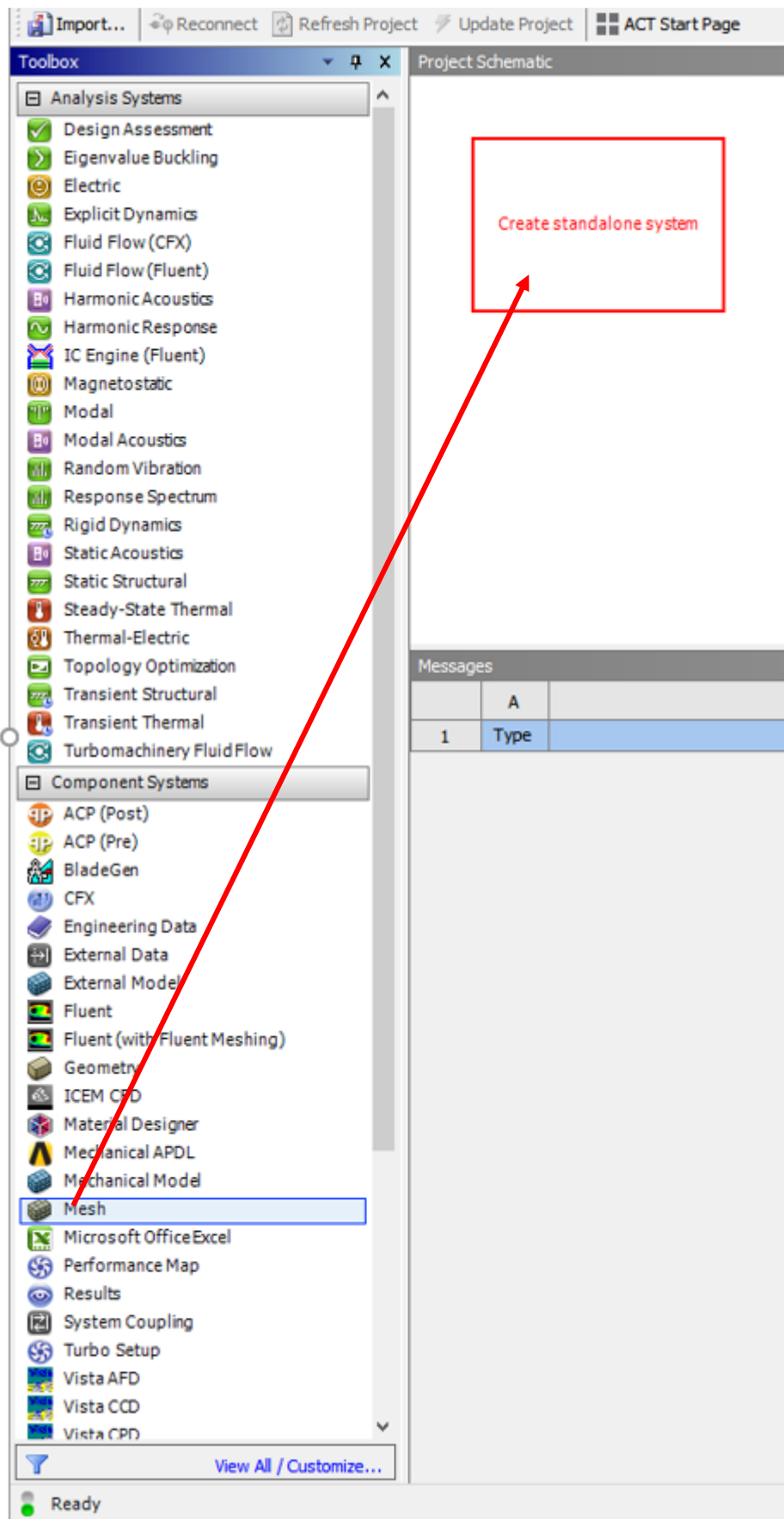
- 1) Open Ansys Workbench and save project as ExNo4 in folder ExNo4 (*File->Save As*).

RULE OF THUMB NO 1: *Create a separate catalog for each project*

RULE OF THUMB NO 2: In the names of directories do not use: spaces, special characters (*e.g. @#\$\$%^&* itp.*) and polish marks



- 2) Select the *Mesh* module and open *Spaceclaim*. To do this, hold the left mouse button (LMB) on the *Mesh* module and drag it to the *Project Schematic* field. Then double-click LMB on *Geometry* to start the *Spaceclaim* program in which the geometry will be created. Note that in the lower left corner of the screen there is an inscription informing what program is running.



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

Toolbox

Analysis Systems

- Design Assessment
- Eigenvalue Buckling
- Electric
- Explicit Dynamics
- Fluid Flow (CFX)
- Fluid Flow (Fluent)
- Harmonic Acoustics
- Harmonic Response
- IC Engine (Fluent)
- Magnetostatic
- Modal
- Modal Acoustics
- Random Vibration
- Response Spectrum
- Rigid Dynamics
- Static Acoustics
- Static Structural
- Steady-State Thermal
- Thermal-Electric
- Topology Optimization
- Transient Structural
- Transient Thermal
- Turbomachinery Fluid Flow

Component Systems

- ACP (Post)
- ACP (Pre)
- BladeGen
- CFX
- Engineering Data
- External Data
- External Model
- Fluent
- Fluent (with Fluent Meshing)
- Geometry
- ICEM CFD
- Material Designer
- Mechanical APDL
- Mechanical Model
- Mesh
- Microsoft Office Excel
- Performance Map
- Results
- System Coupling
- Turbo Setup
- Vista AFD
- Vista CCD
- Vista CPD

Project Schematic

A

1 Mesh

2 Geometry ?

3 Mesh ?

Mesh

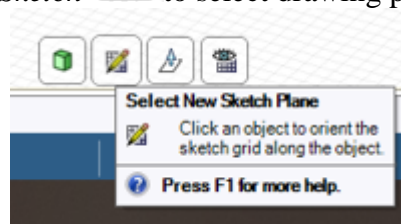
Messages

	A	
1	Type	

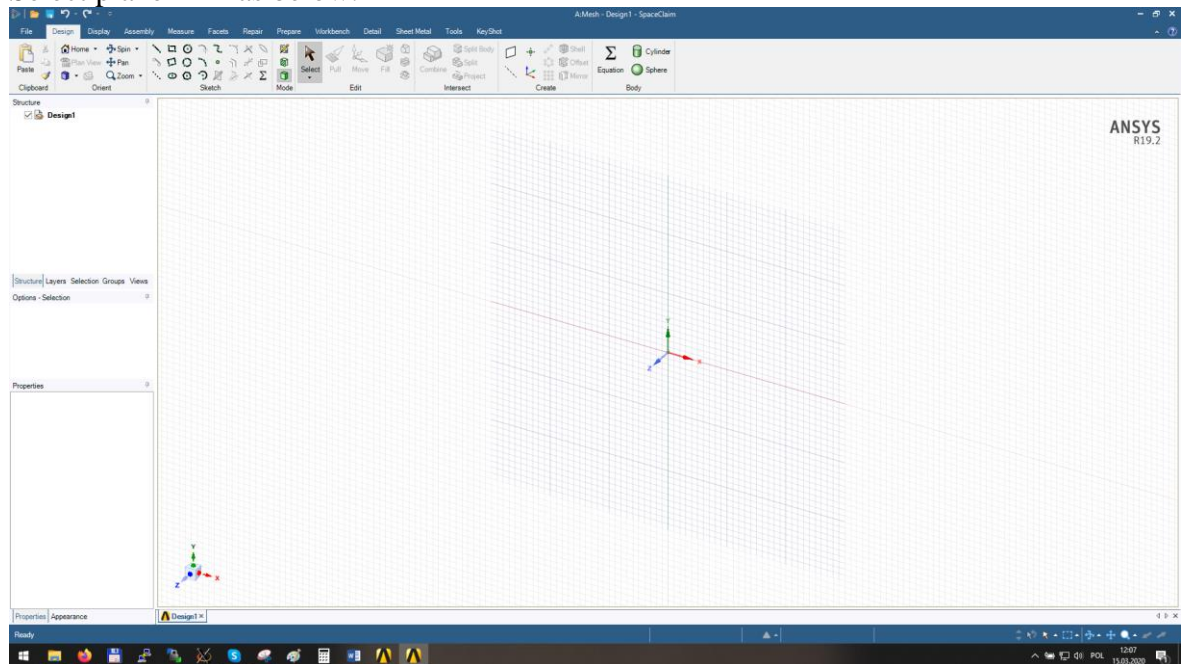
View All / Customize...


Starting SpaceClaim...

- 3) Click LMB *Select New Sketch*  to select drawing plane.




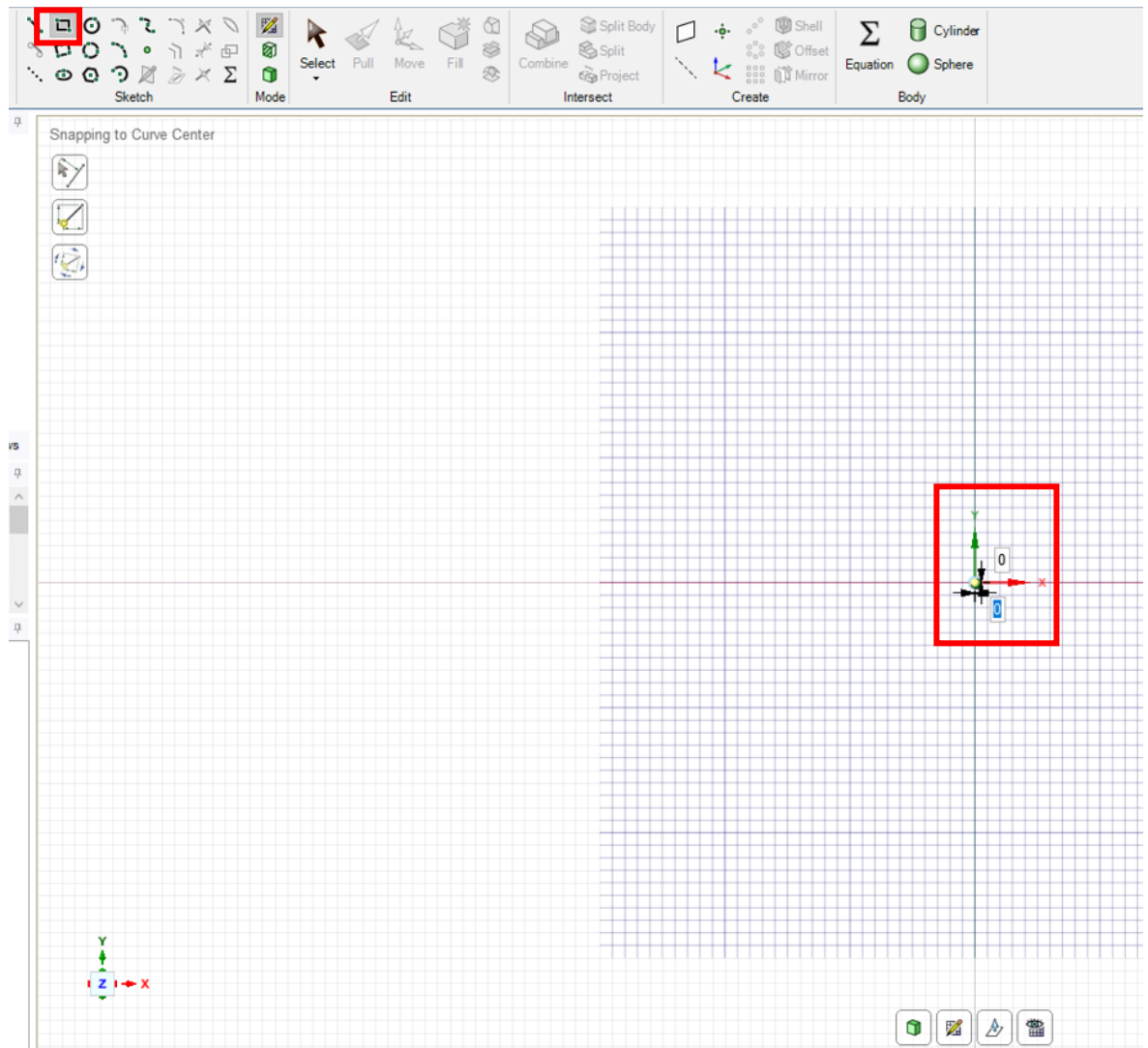
Select plane X-Y as below.



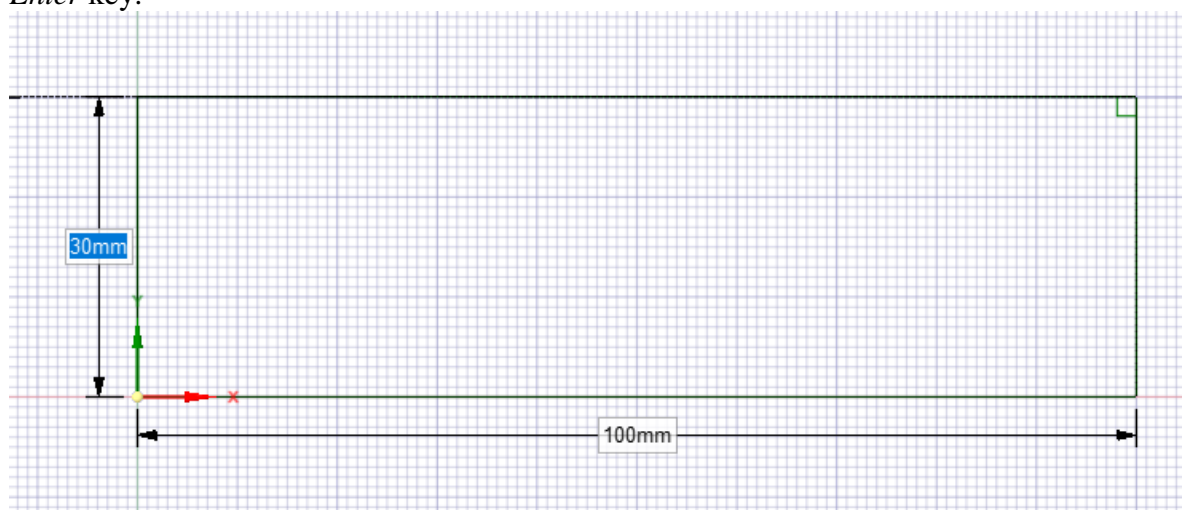
- 4) Click *Plan View*  to rotate the drawing plane parallel to the screen (you can also do this by pressing *Shift + v*).



- 5) In the panel at the top of the screen, select the draw rectangle icon  and move the cursor to the center of the coordinate system and then press the LMB key.



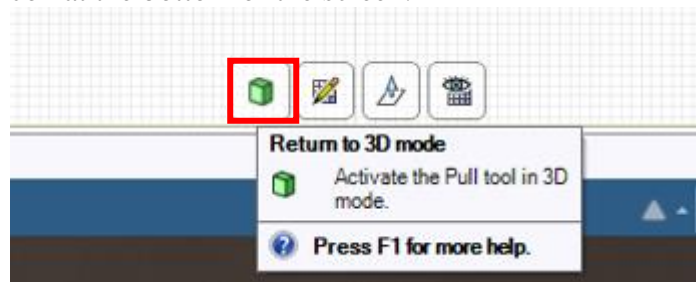
You can switch between dimension edit fields with the *Tab* key. Set the horizontal dimension to 100 and the vertical dimension to 30 mm and press the *Enter* key.



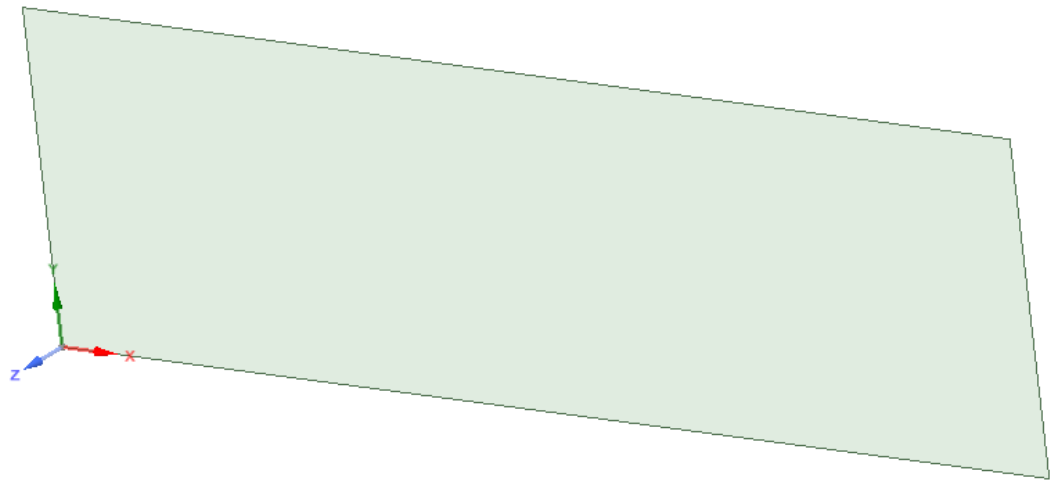
Remember, that if you fail at any time you can click the undo icon (located in the upper left corner of the screen) or *Ctrl* + *Z*.



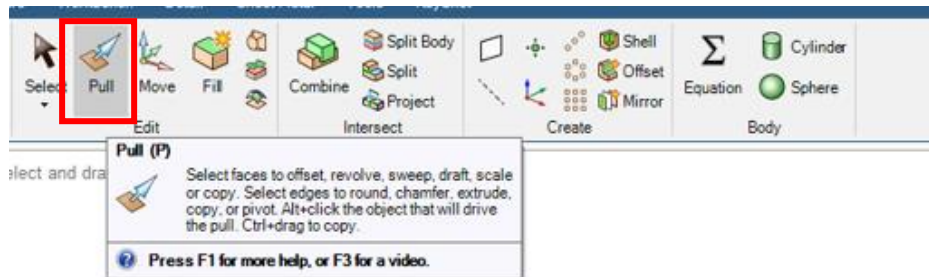
- 6) To exit the rectangle drawing command, press *Esc* and LMB, click the *Return to 3D mode* icon at the bottom of the screen.



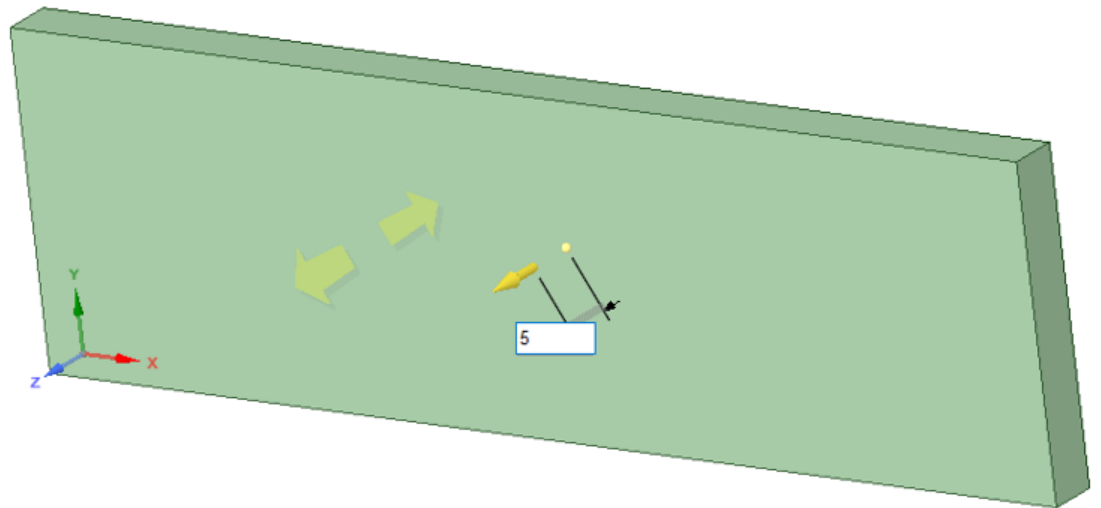
- 7) Rotate the view by holding down the *Scroll* mouse button and moving it to get an isometric view similar to the one below.



- 8) Select *Pull*



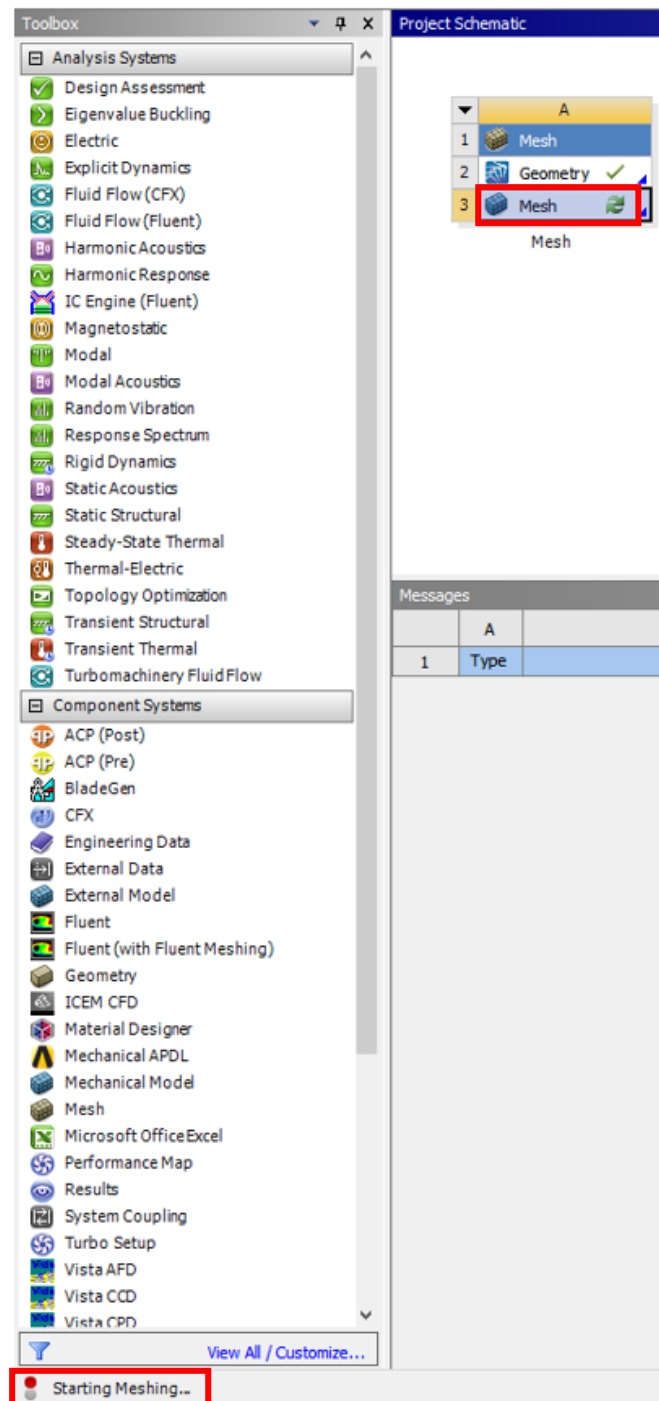
Then position the cursor as shown below. By moving the cursor while pressing LMB you will notice the changing dimension of the length. Enter 5 mm and confirm with Enter (you may have to do it while holding LMB).



9) Close *Spaceclaim* save project in *Workbench* using *Ctrl + s*.



2.2. NUMERICAL MESH









1) Open *Ansys Meshing* by double-click LMB on *Mesh*



- 2) In Ansys Meshing: 1) click Mesh, 2) Change field *Physisc Preference* into *CFD*,
3) Change *Solver Preference* into *CFX*, 4) Click LMB *Generate Mesh*

A : Mesh - Meshing [ANSYS Academic Research Mechanical and CFD]

File Edit View Units Tools Help |  **Generate Mesh** 

        **4**

Show Vertices Close Vertices 1,e-004 (Auto) Wireframe








Size Location Convert Miscellaneous Tolerances

Reset Explode Factor: Assembly Center

Mesh Update Mesh Mesh Control Mesh Edit Metric G

Outline

Filter: Name

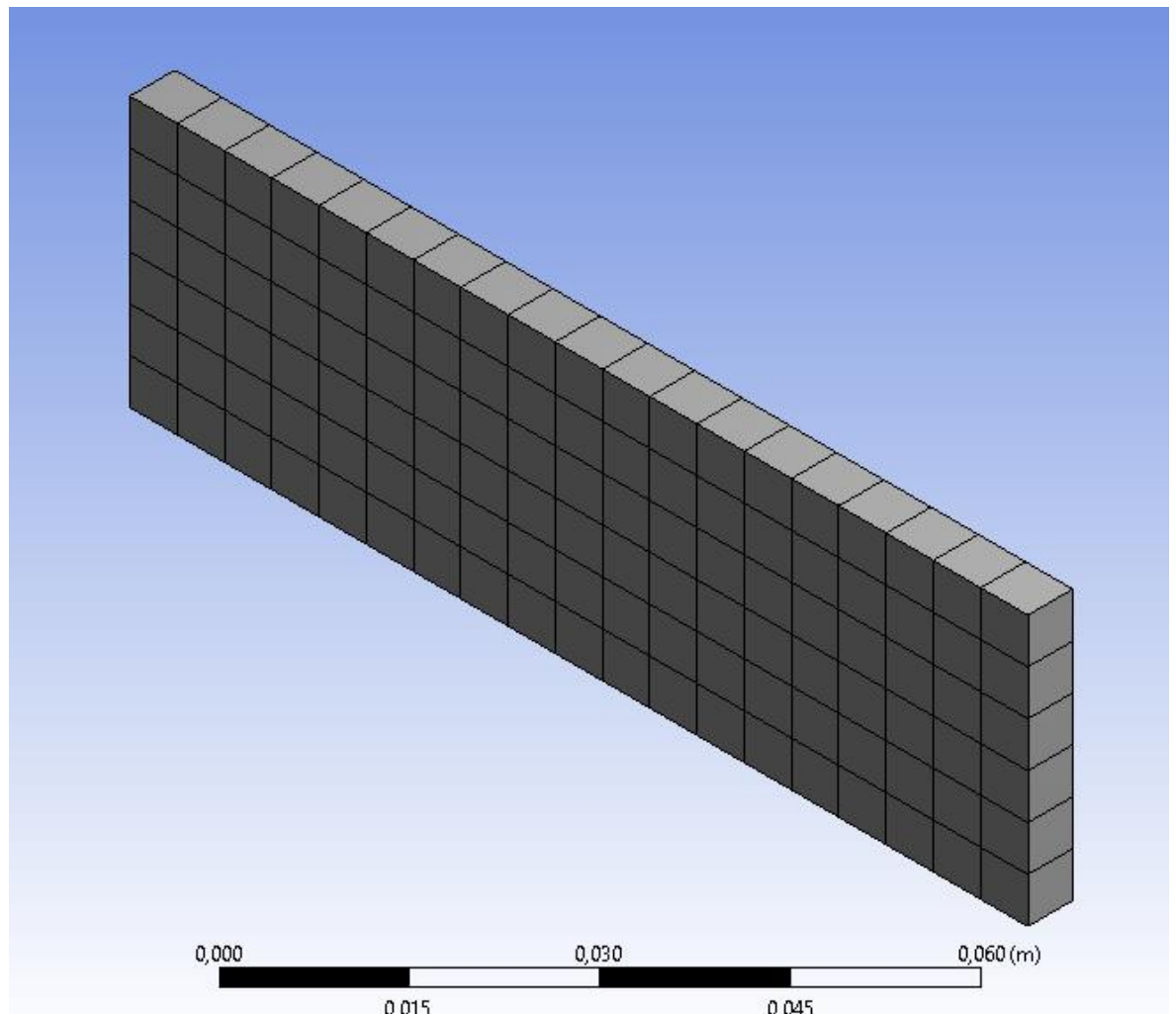
Project

- Model (A3)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh** **1**

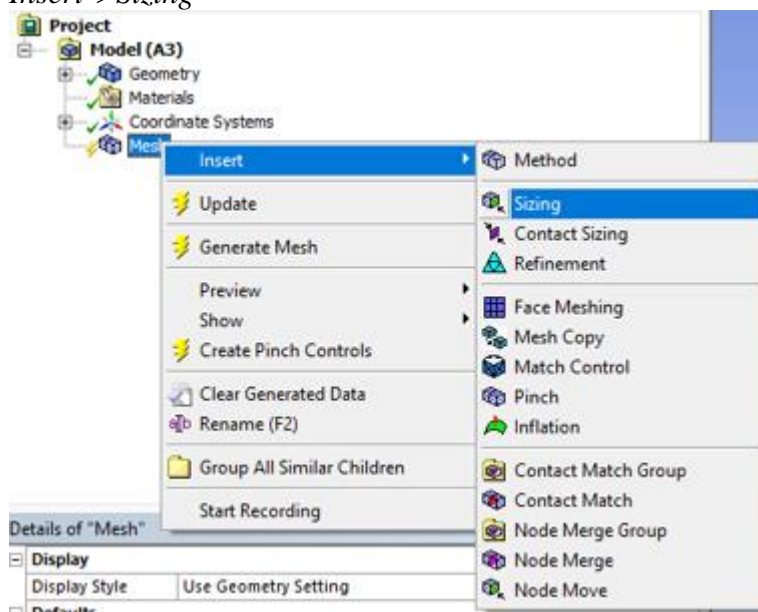
Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD 2
Solver Preference	CFX 3
Element Order	Linear
<input type="checkbox"/> Element Size	Default (5,1264e-003 m)
Sizing	
Quality	
Inflation	
Advanced	
Statistics	

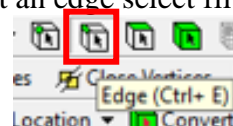
The default grid is not valid. The grid should be edited.



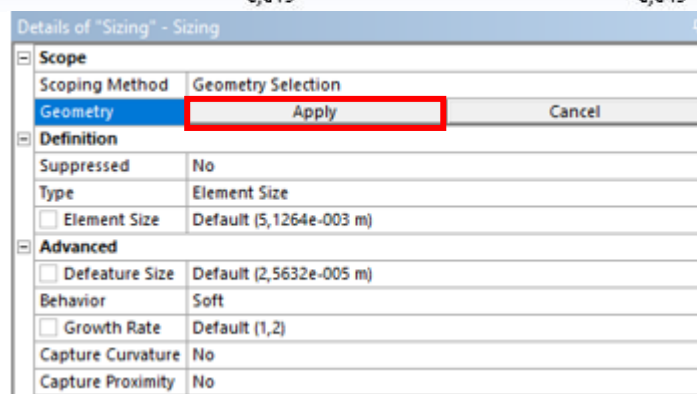
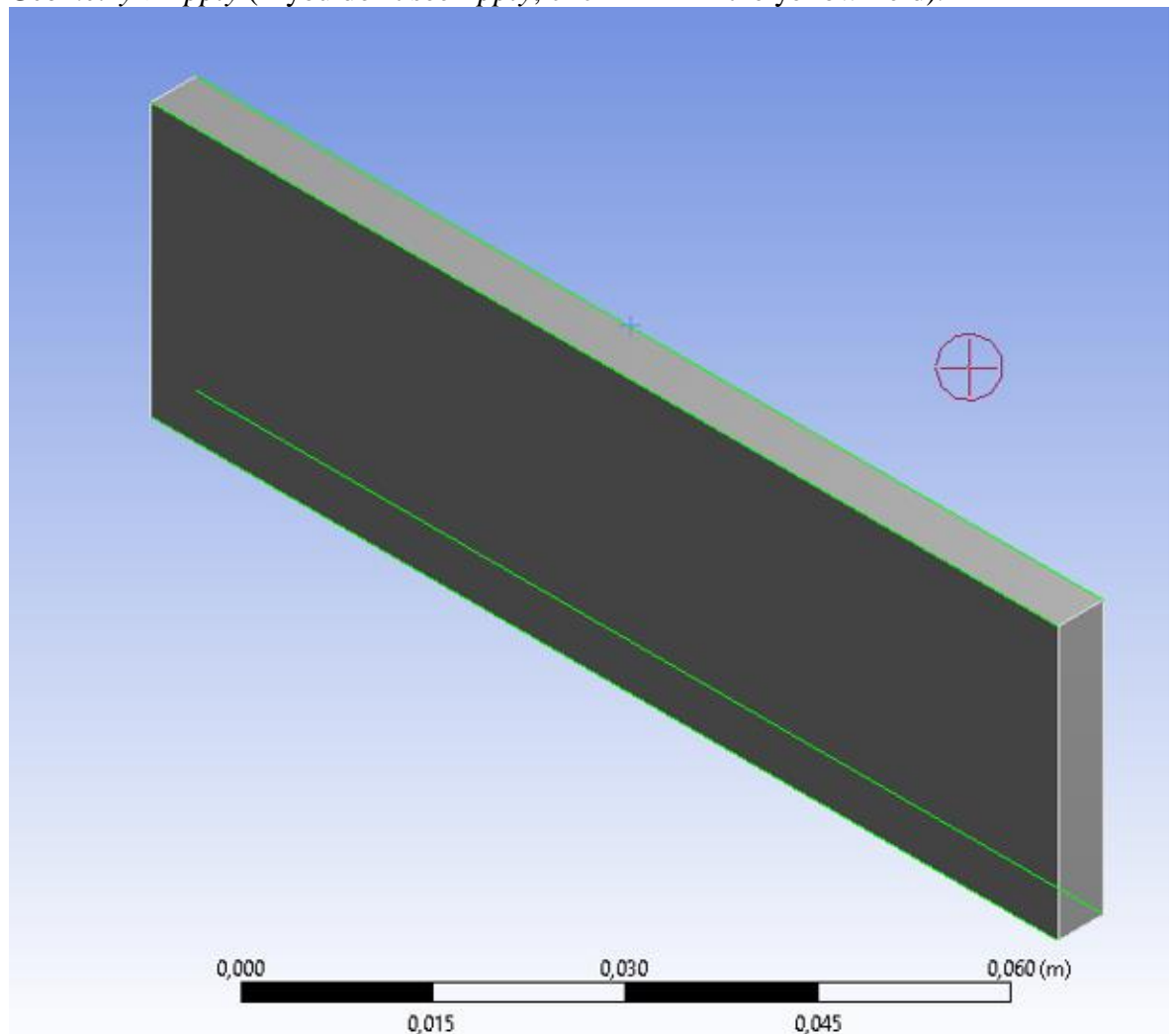
- 3) In *Ansys Meshing* press the right mouse button (RMB) on *Mesh* and select *Insert->Sizing*



At the top of the screen, select an edge select filter *Edge*



While holding down the *Ctrl* key, select 4 edges as shown below and confirm *Geometry->Apply* (if you don't see *Apply*, click LMB in the yellow field).

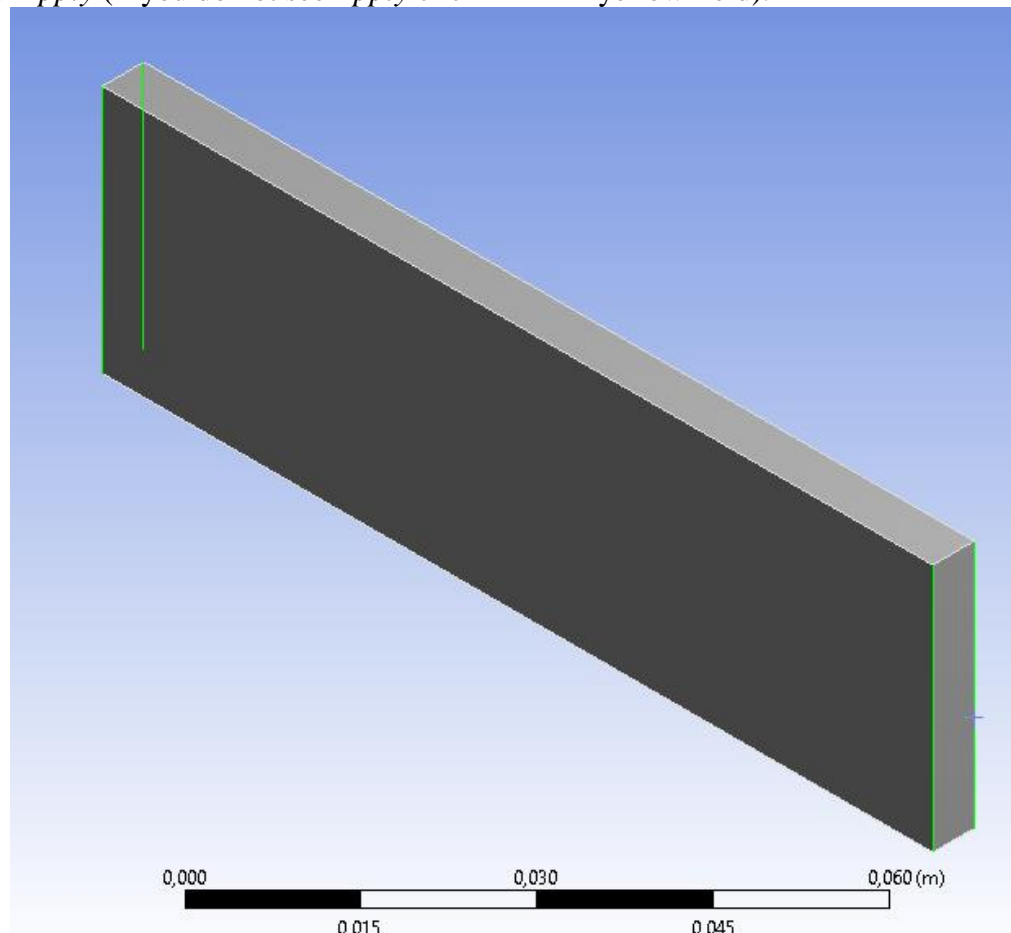


Change *Definition* as in the figure below

Details of "Edge Sizing" - Sizing	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
[-] Definition	
Suppressed	No
Type	Number of Divisions
<input checked="" type="checkbox"/> Number of Divisions	100
[-] Advanced	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

Click *Generate Mesh* and check generated mesh (if you can't see the mesh click LMB on *Mesh* in the tree on the left).

- 4) In *Ansys Meshing* press RMB on *Mesh* and select *Insert->Sizing*. While holding down the *Ctrl* key, select 4 edges as in the figure below and confirm *Geometry->Apply* (if you do not see *Apply* click LMB in yellow field).



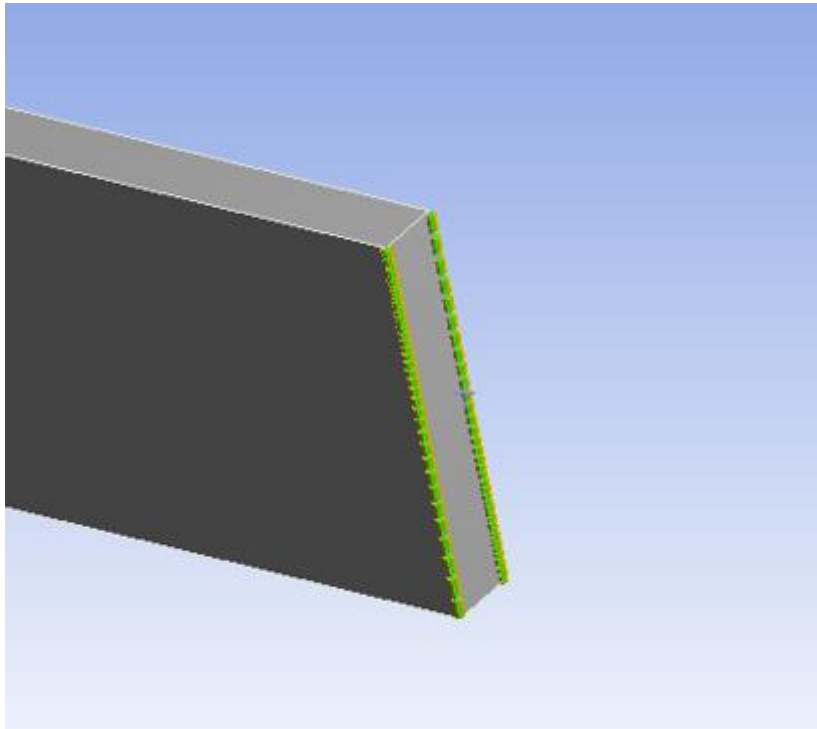
Apply below settings

Details of "Edge Sizing 2" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	50
Advanced	
Size Function	Uniform
Behavior	Hard
Bias Type	- - - - -
Bias Option	Bias Factor
<input type="checkbox"/> Bias Factor	20,0
Reverse Bias	No Selection

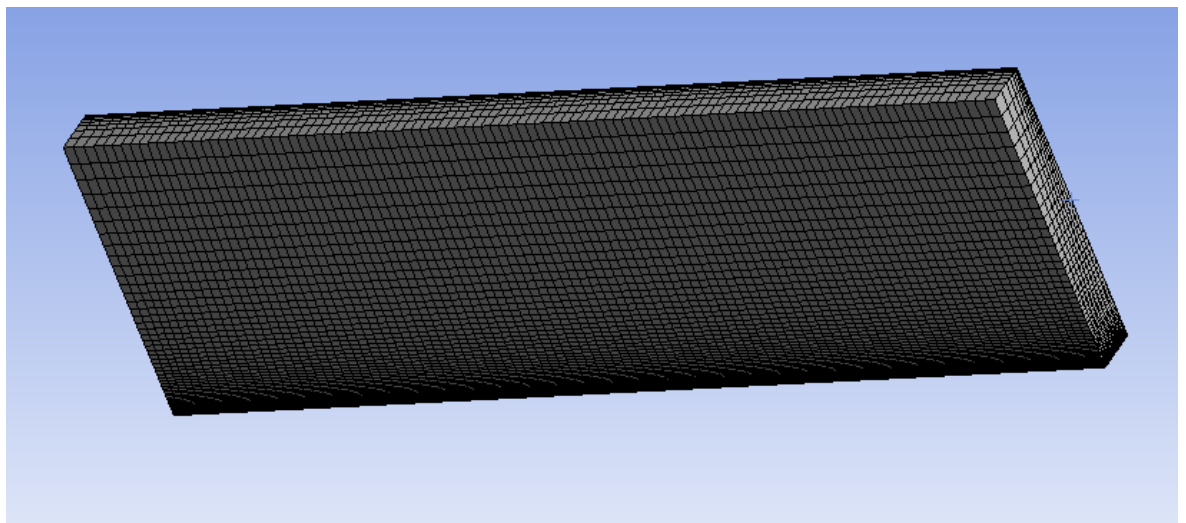
Click *Generate Mesh* and check generated mesh (if you can't see the mesh click LMB on *Mesh* in the tree on the left). The grid is not symmetrical. To change this, select the LMB option *Reverse Bias*

Details of "Edge Sizing 2" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	50
Advanced	
Size Function	Uniform
Behavior	Hard
Bias Type	- - - - -
Bias Option	Bias Factor
<input type="checkbox"/> Bias Factor	20,0
Reverse Bias	No Selection

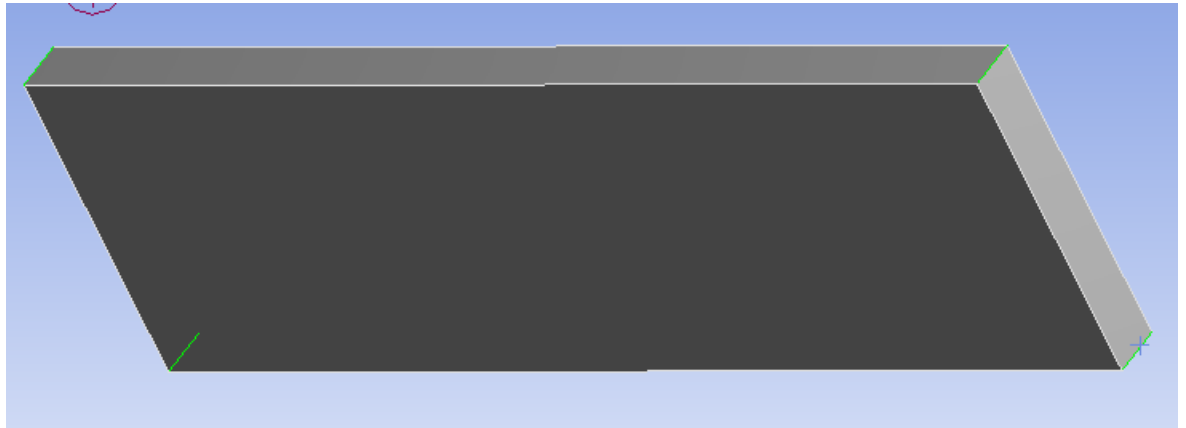
Then pick 2 edges as in the figure below and confirm *Apply*.



Click *Generate Mesh* and check the generated mesh (if you can't see the mesh click LMB on *Mesh* in the tree on the left).



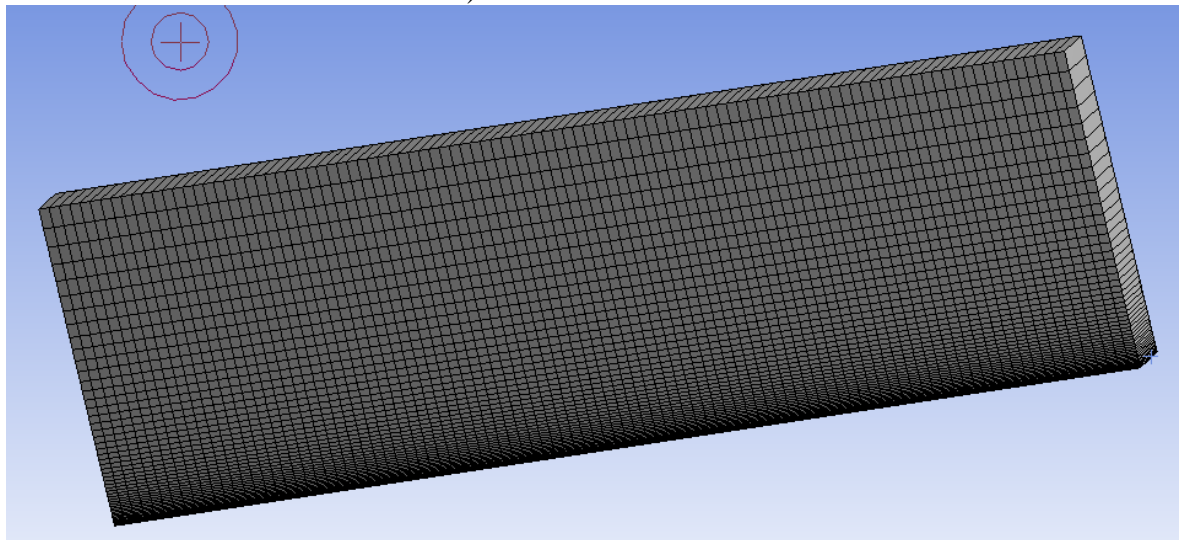
- 5) In *Ansys Meshing* RMB on *Mesh* and select *Insert->Sizing*. With the *Ctrl* key held down, select 4 edges as in the figure below and confirm *Geometry->Apply* (if you do not see *Apply* click LMB in yellow field).



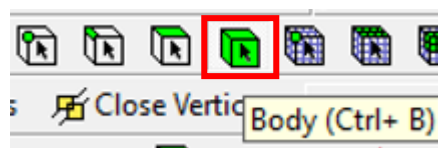
Apply below settings

Details of "Edge Sizing 3" - Sizing	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
[-] Definition	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	1
[-] Advanced	
Size Function	Uniform
Behavior	Hard
Bias Type	No Bias

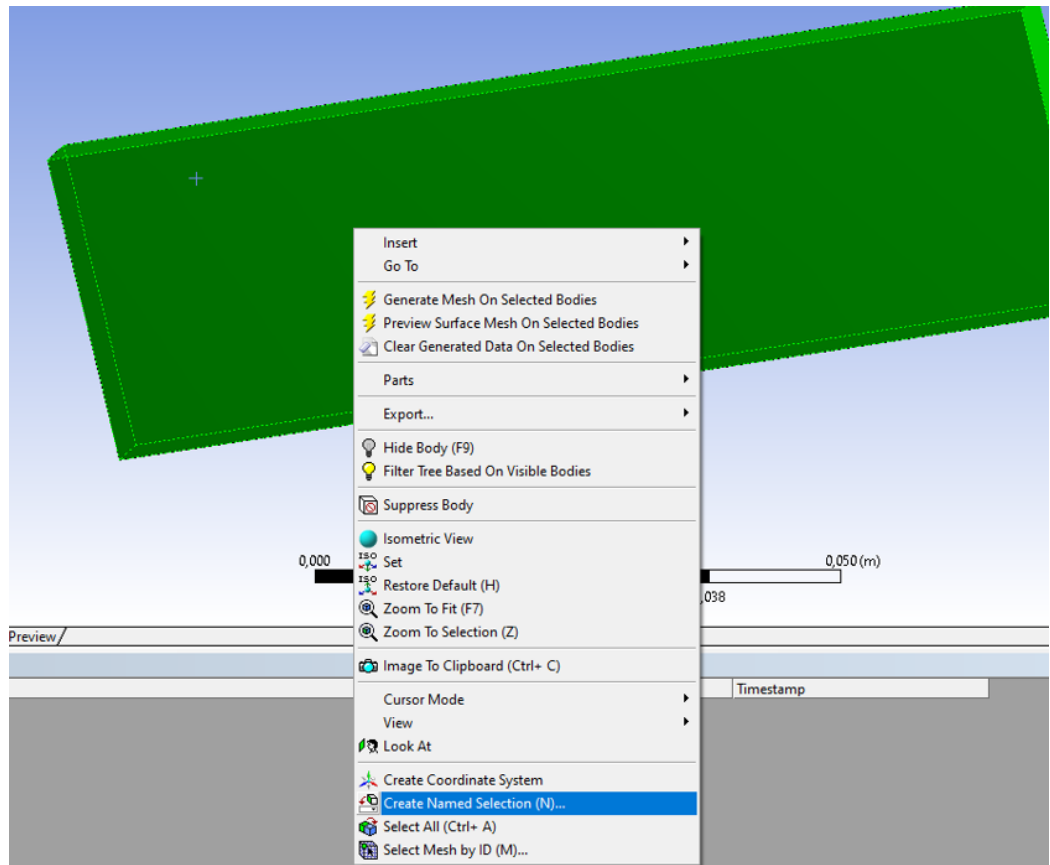
Click *Generate Mesh* and check the generated mesh (if you can't see the mesh LMB on *Mesh* in the tree on the left).



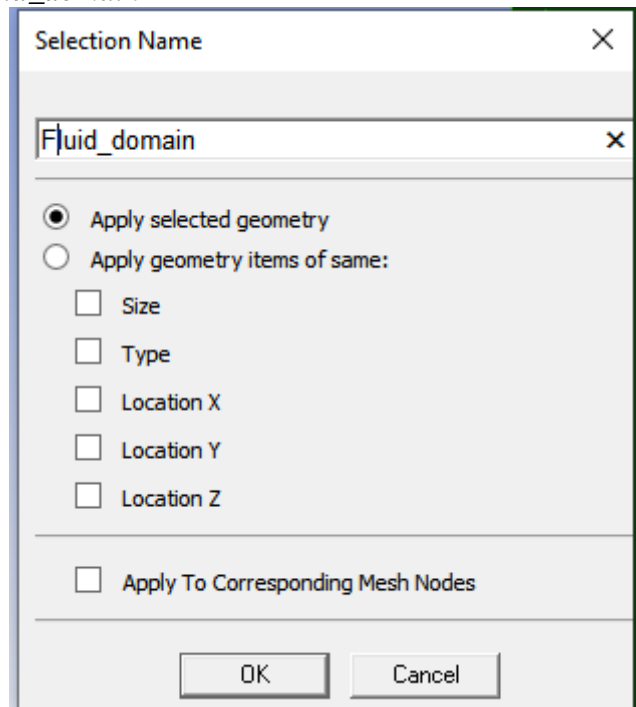
- 6) The last step is to name the volumes and surfaces. Select the LMB body selection filter



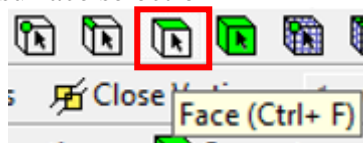
Choose a cuboid LMB, and then click RMB and choose *Create Named Selection*



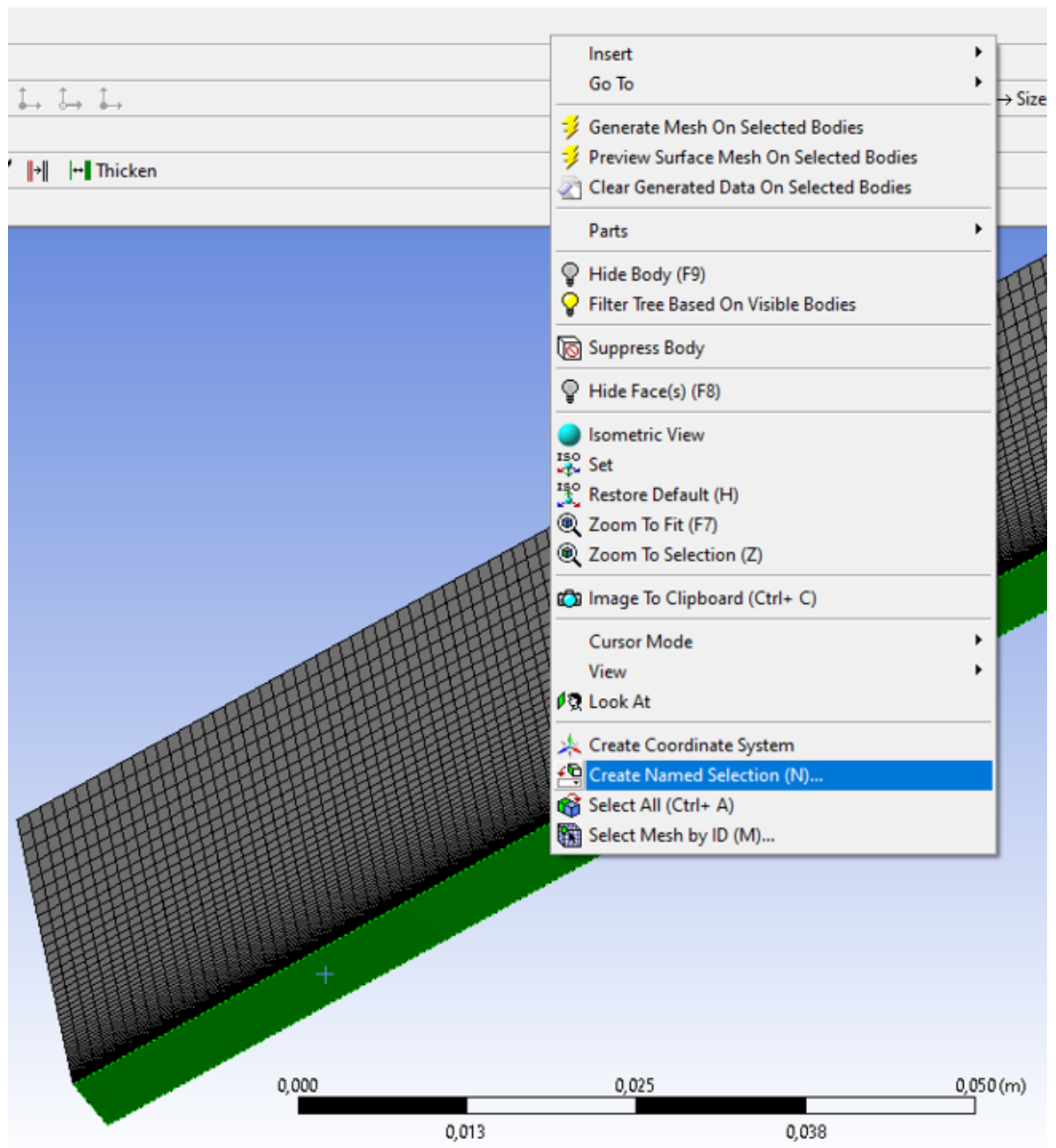
Name it as *Fluid_domain*



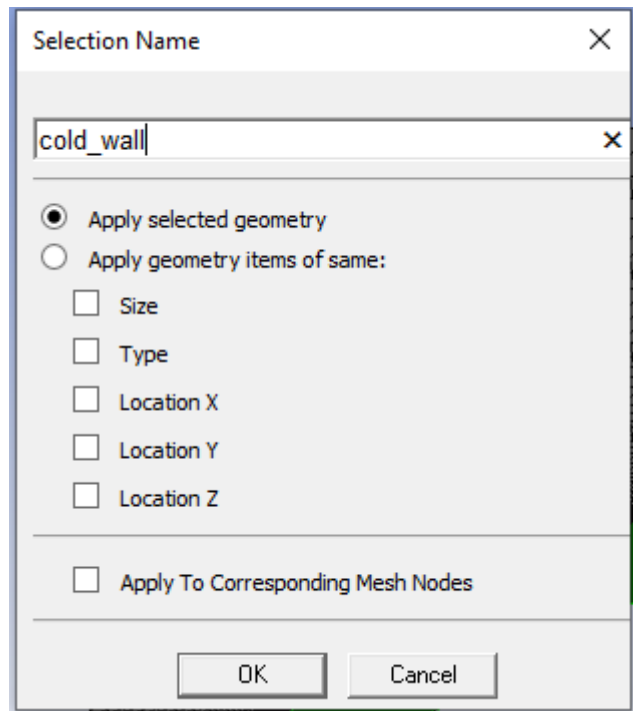
7) Then change the filter to surface selection



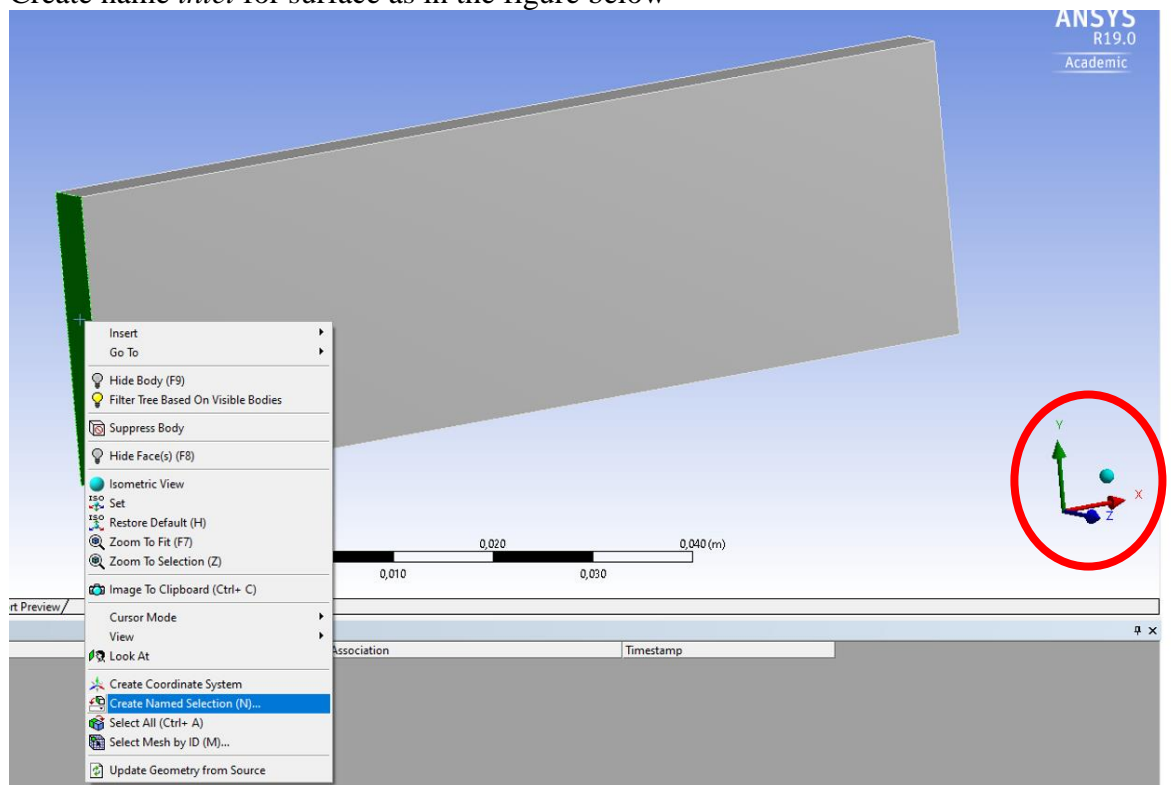
LMB indicate the outer surface at which the mesh is dense and Click RMB, then select *Create Named Selection*

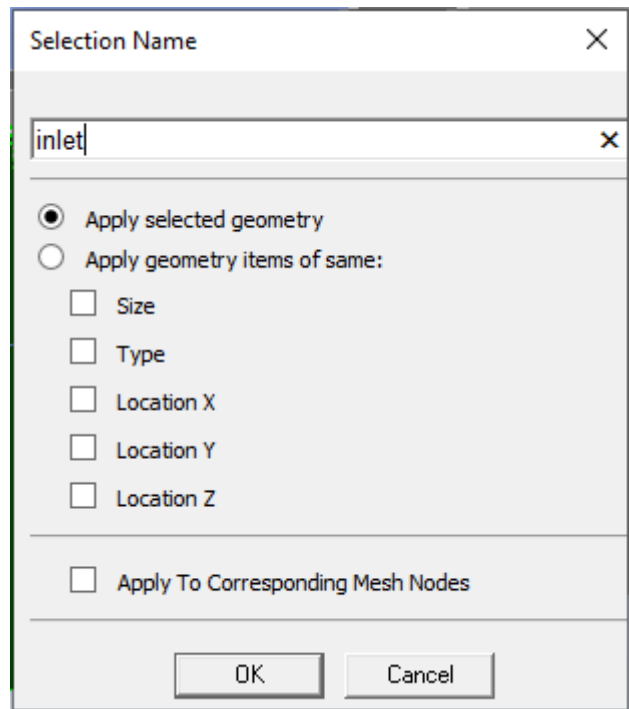


Call it *cold_wall*

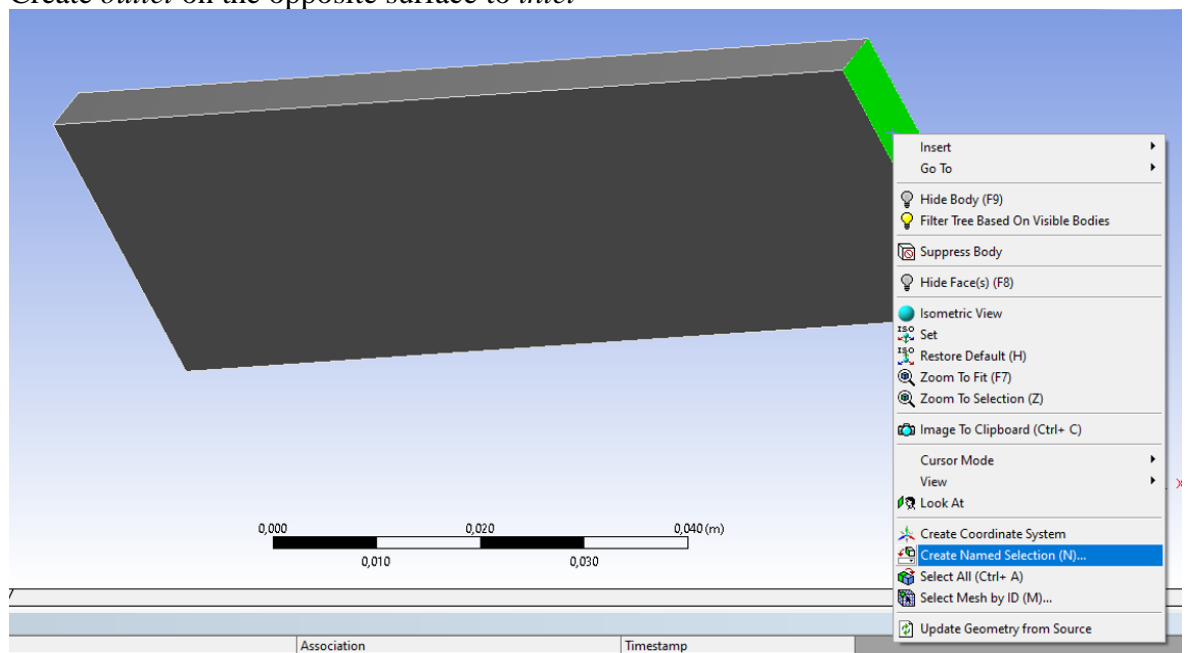


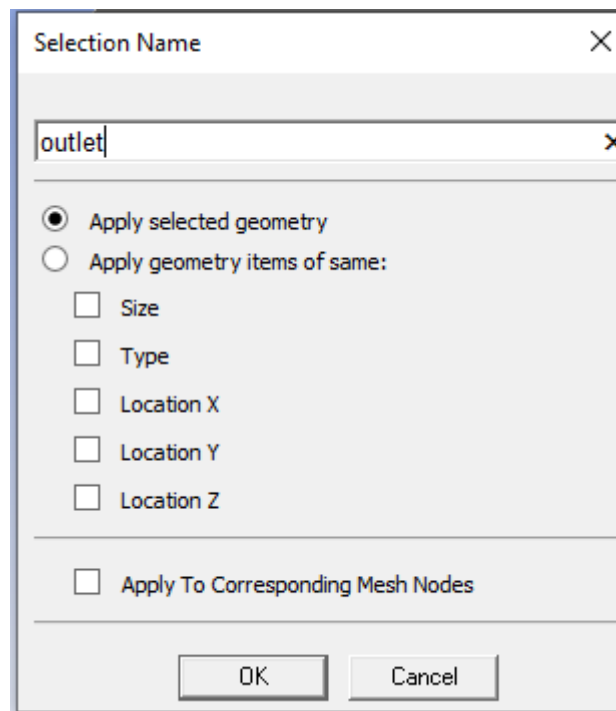
8) Create name *inlet* for surface as in the figure below



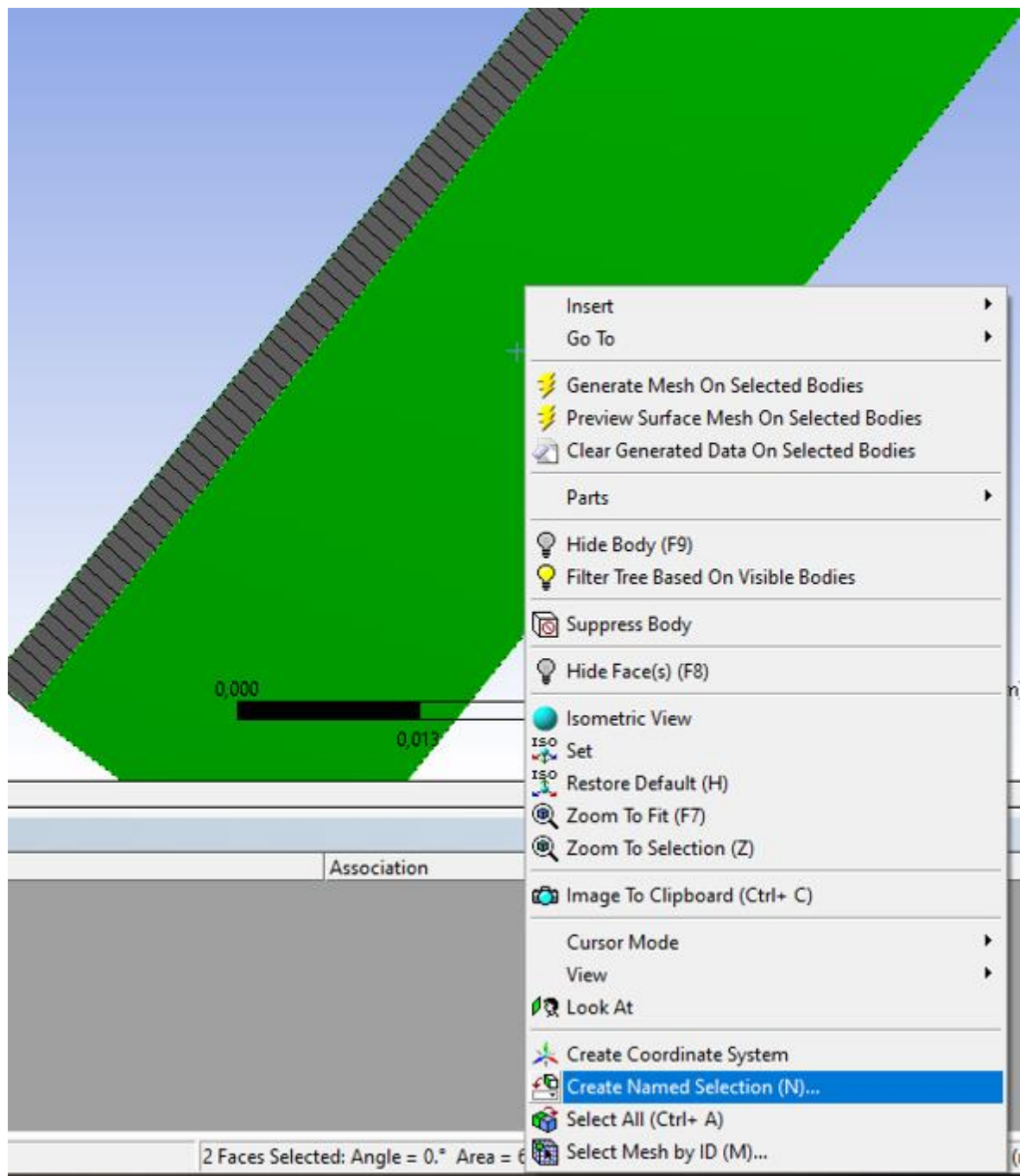


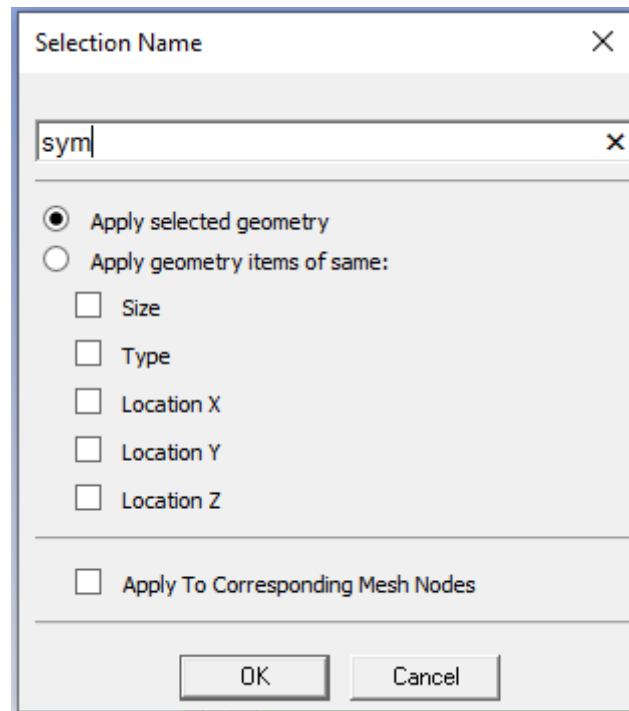
9) Create *outlet* on the opposite surface to *inlet*





10) Similarly, create a *sym* name for two large flat surfaces





A screenshot of a 'Selection Name' dialog box. The title bar at the top says 'Selection Name' with a close button (X) on the right. Below the title bar is a text input field containing the text 'sym'. Underneath the text field are two radio button options: 'Apply selected geometry' (which is selected) and 'Apply geometry items of same:'. Below the second radio button are five unchecked checkboxes: 'Size', 'Type', 'Location X', 'Location Y', and 'Location Z'. At the bottom of the dialog, there is another unchecked checkbox labeled 'Apply To Corresponding Mesh Nodes'. At the very bottom are two buttons: 'OK' and 'Cancel'.

Selection Name

sym

☒ Apply selected geometry

☐ Apply geometry items of same:

☐ Size

☐ Type

☐ Location X

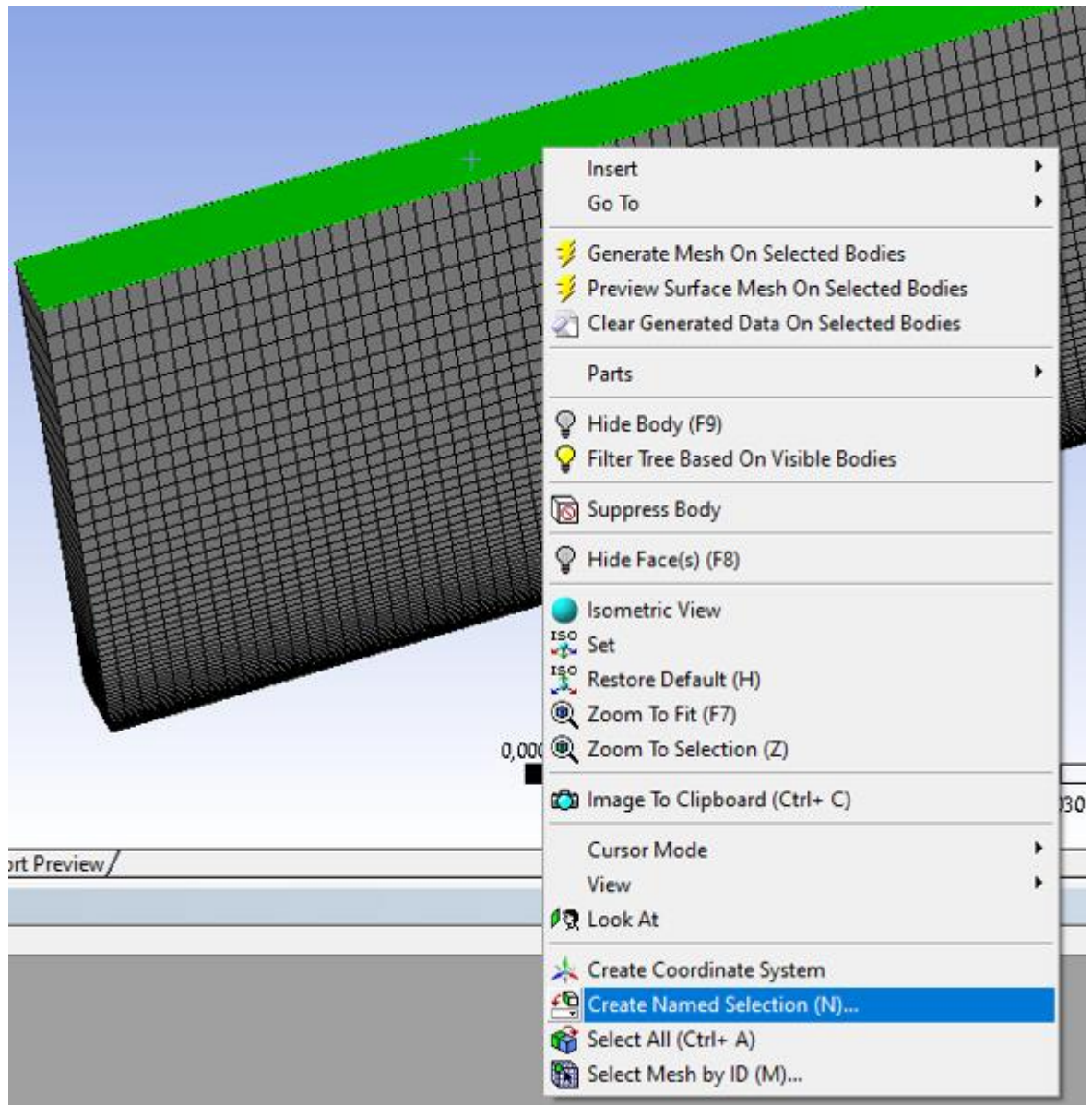
☐ Location Y

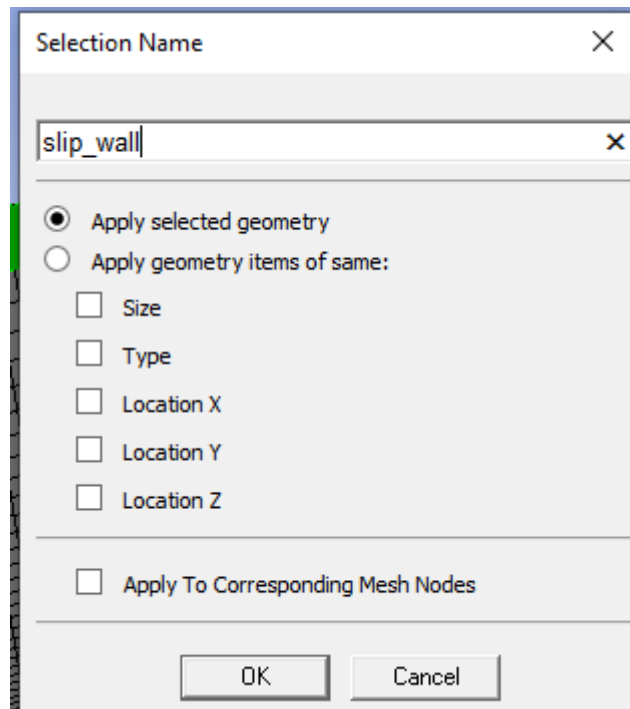
☐ Location Z

☐ Apply To Corresponding Mesh Nodes

OK Cancel

11) Give the name on the last unnamed surface as *slip_wall*

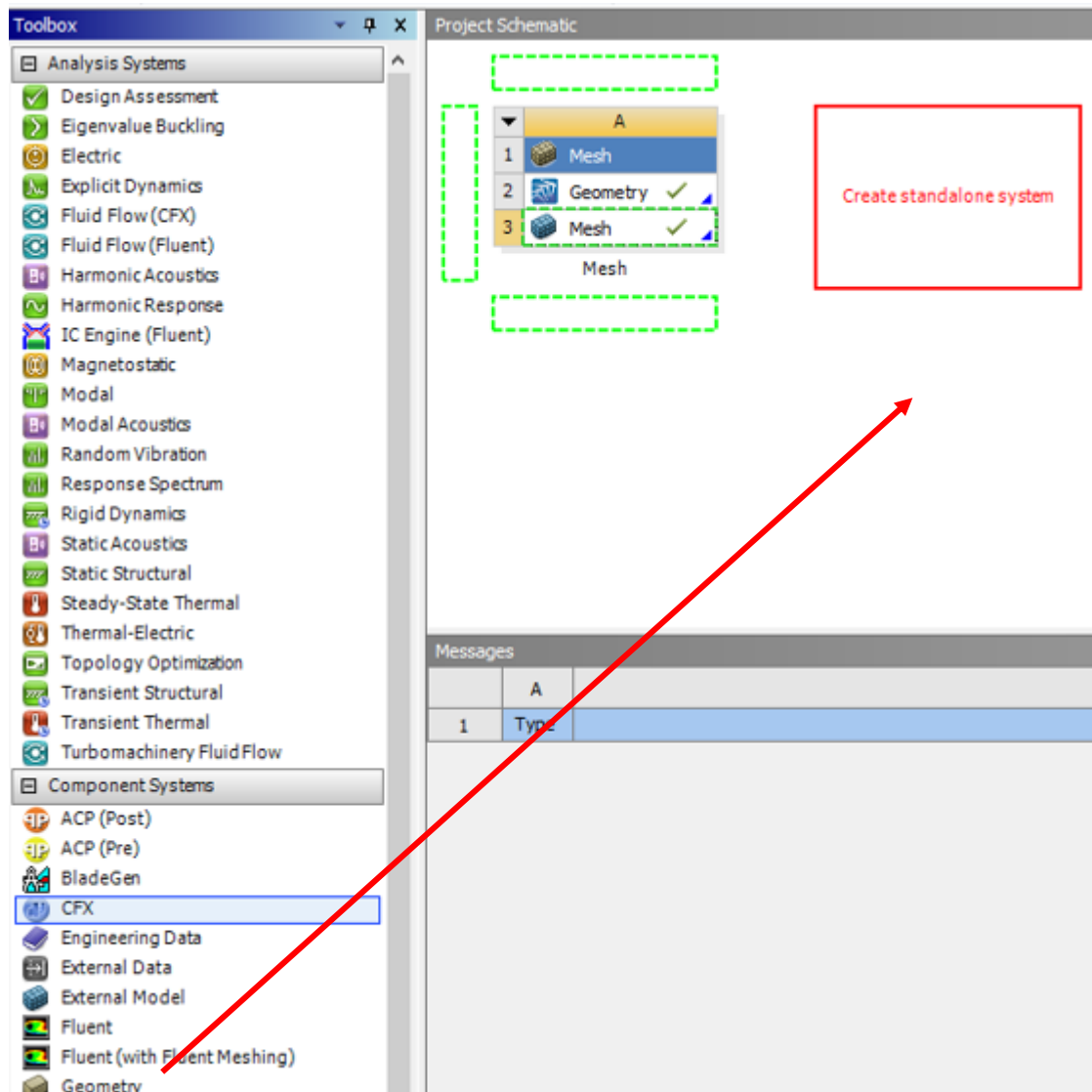




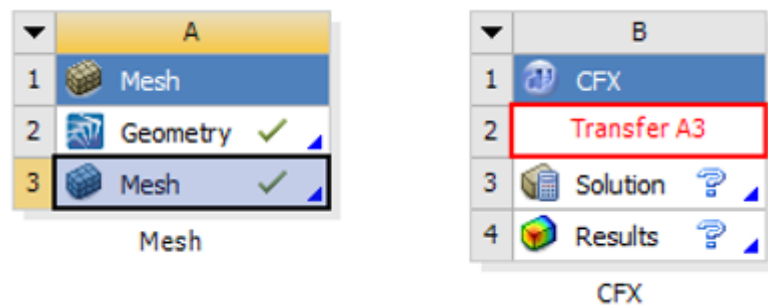
12) Close *Ansys Meshing* and save project in *Workbench*.

2.3. NUMERICAL MODEL

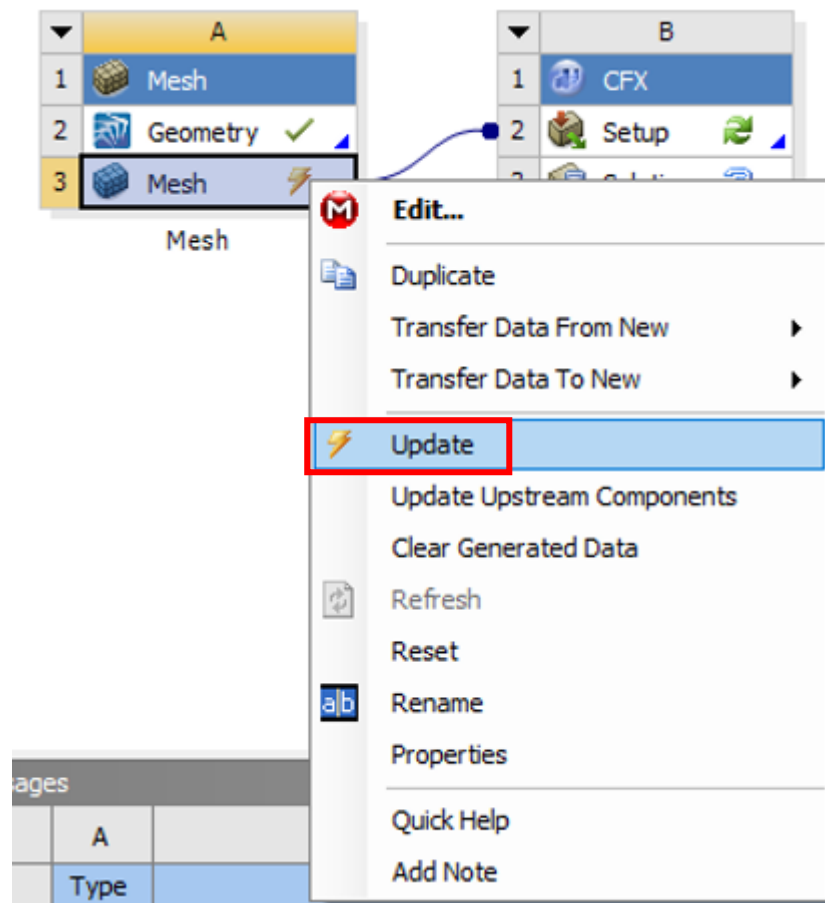
1) Insert *CFX*



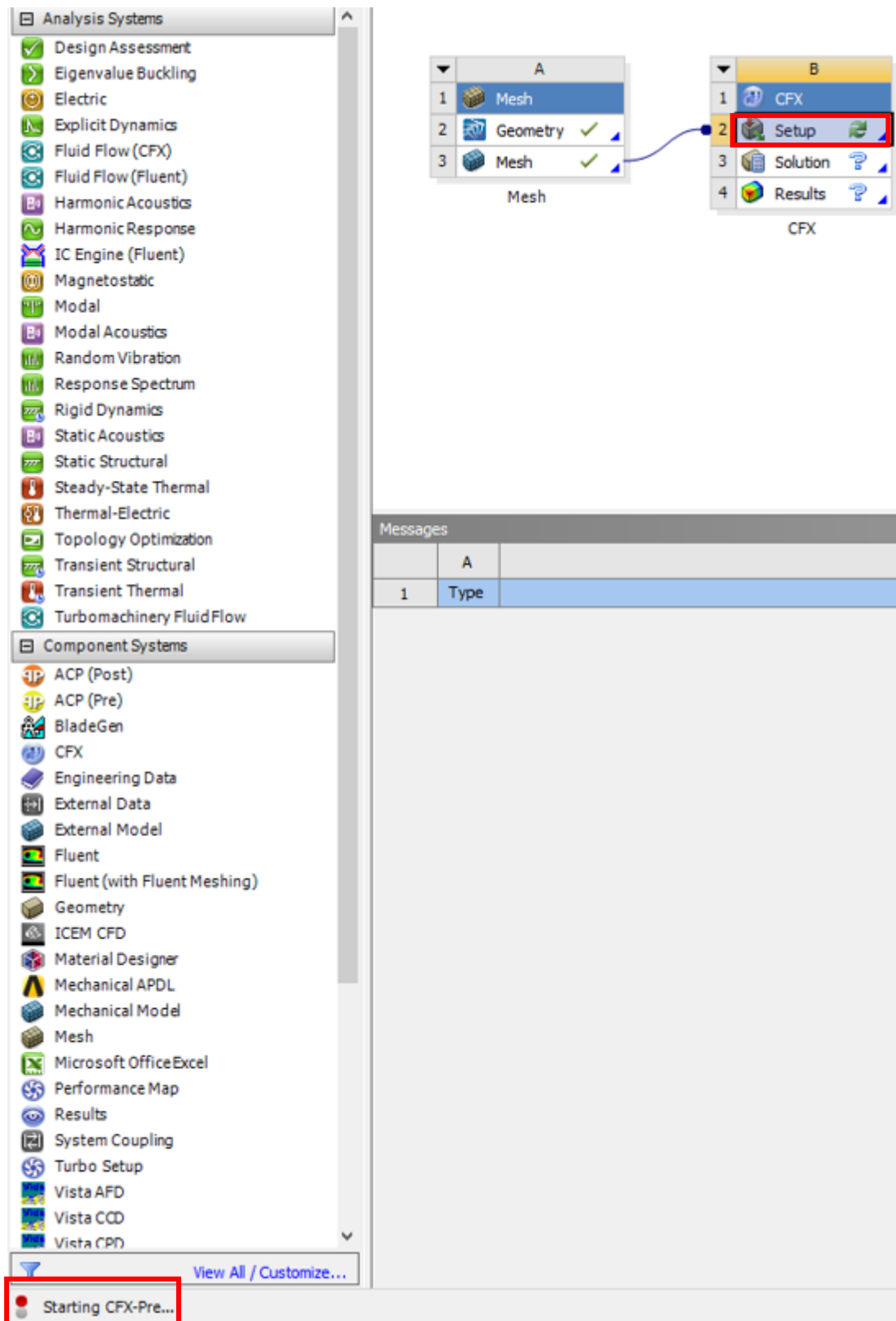
To connect the *Mesh* module with *CFX*, grab LMB *Mesh* (below) and drag it to *Setup* until the *Transfer A3* box appears, then release LMB - the connection has been created



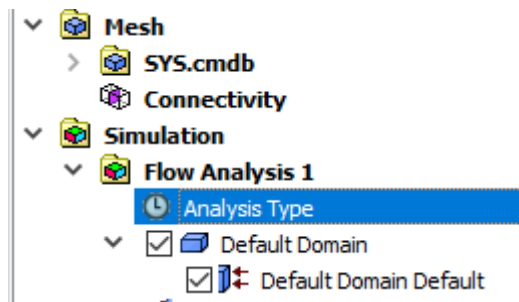
Click RMB on *Mesh* and select *Update*



Double-click on *Setup* to run *Ansys CFX*



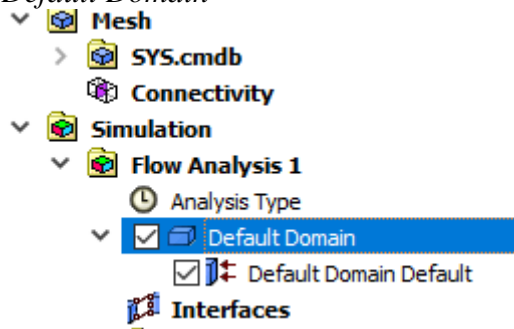
- 2) Open *Analysis Type* by double-click LMB




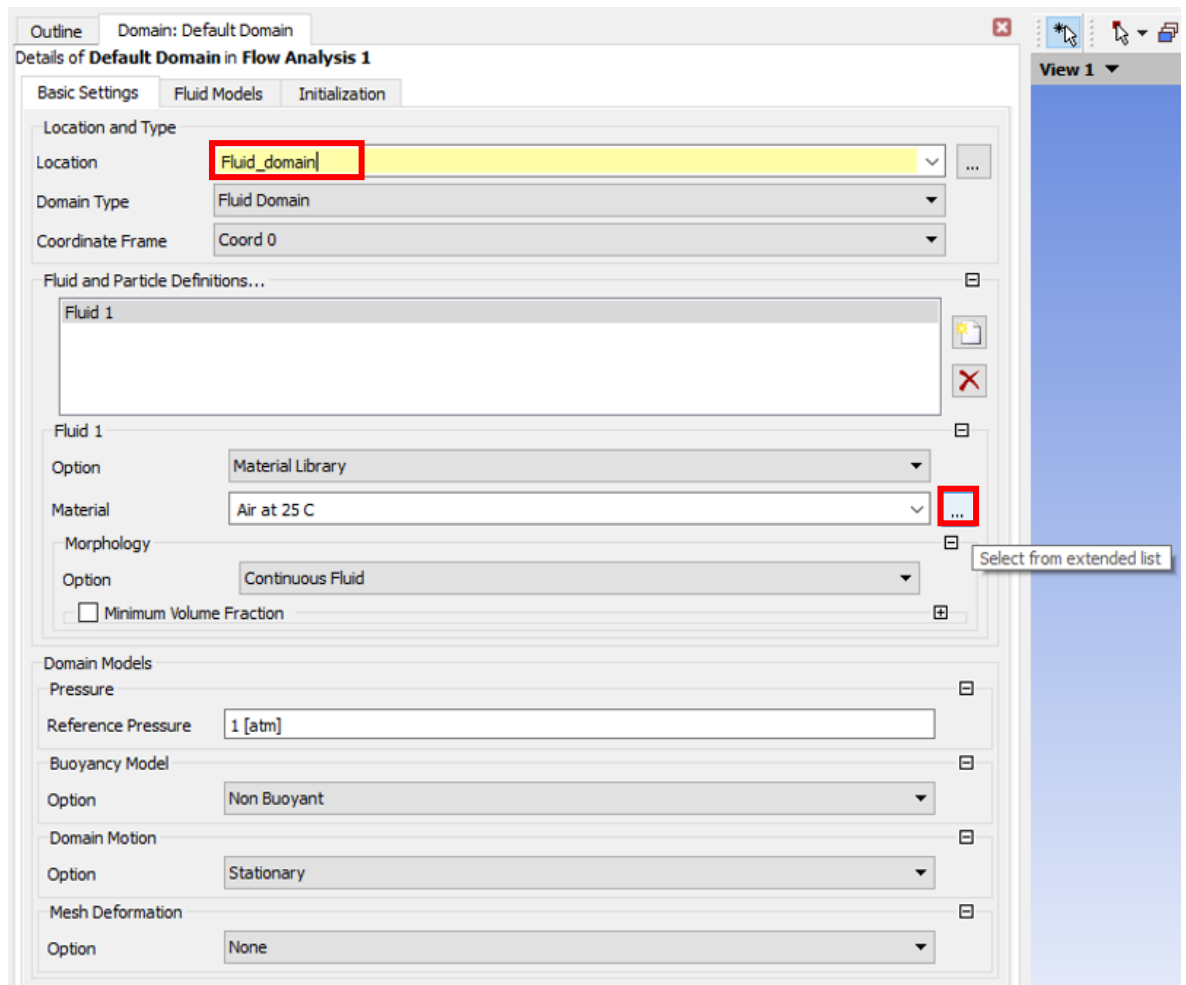
Apply the following settings and confirm *OK*.



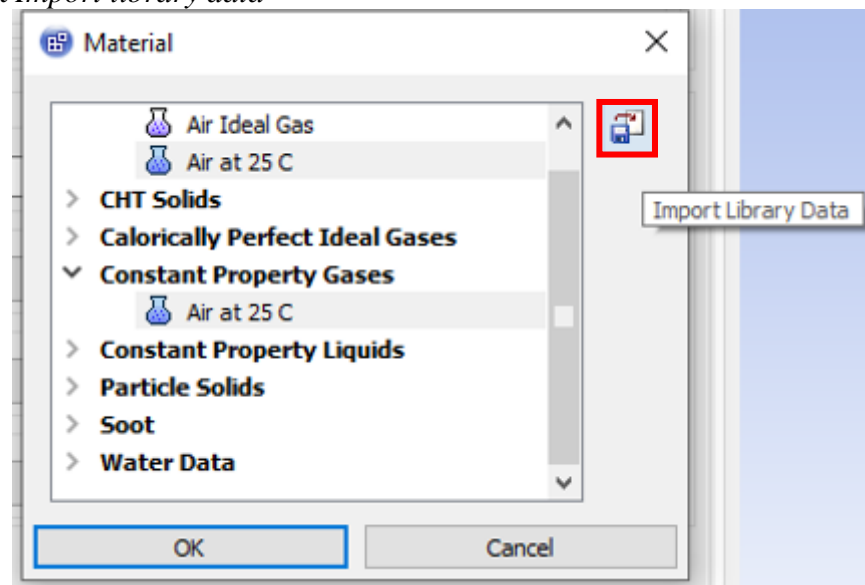
3) Double-click LMB *Default Domain*



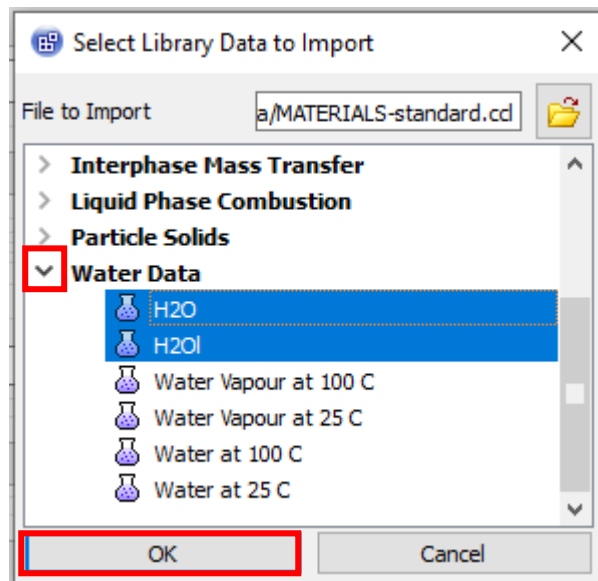
Apply the following settings, select the import material button  from *CFX library*.



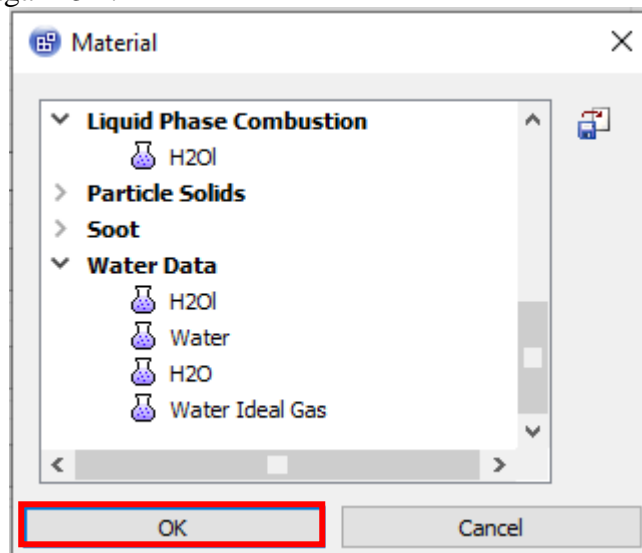
4) Select *Import library data*



5) With the *Ctrl* key pressed, select materials H₂O i H₂O_l and confirm *OK*.



And confirm again *OK*.



6) Close for now *Default Domain*

Outline Domain: Default Domain

Details of **Default Domain** in **Flow Analysis 1**

Basic Settings Fluid Models Initialization

Location and Type

Location Fluid_domain

Domain Type Fluid Domain

Coordinate Frame Coord 0

Fluid and Particle Definitions...

Fluid 1

Fluid 1

Option Material Library

Material Air at 25 C

Morphology

Option Continuous Fluid

☐ Minimum Volume Fraction

Domain Models

Pressure

Reference Pressure 1 [atm]

Buoyancy Model

Option Non Buoyant


Domain Motion

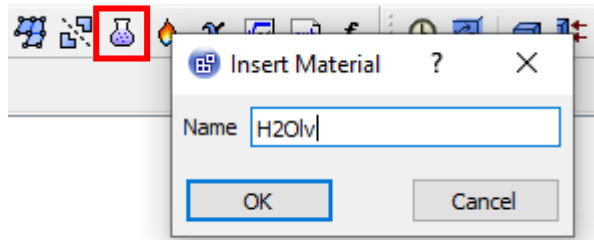
Option Stationary

Mesh Deformation

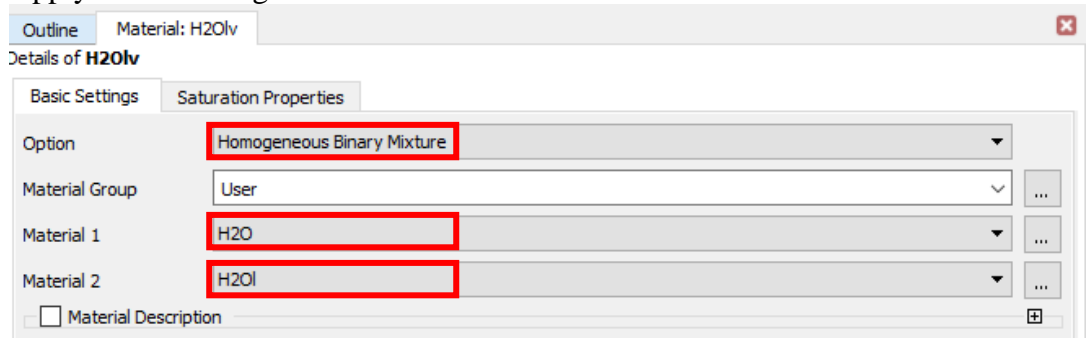
Option None

OK Apply Close

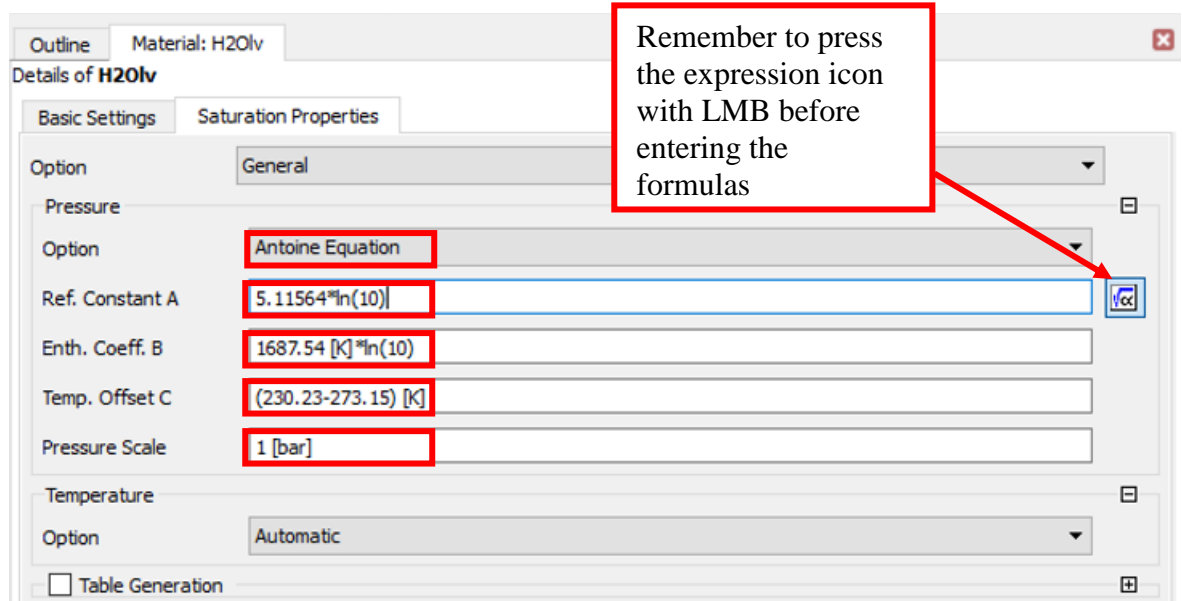
7) Create a new material with the name H2Olv by clicking  icon



8) Apply below settings

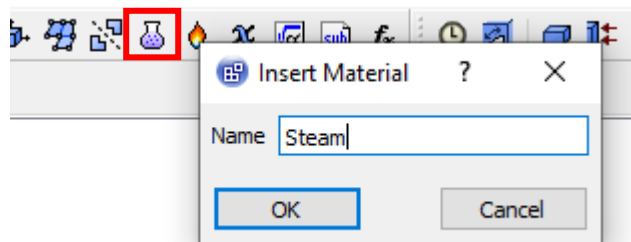


Go to tab *Saturation Properties* and apply as below

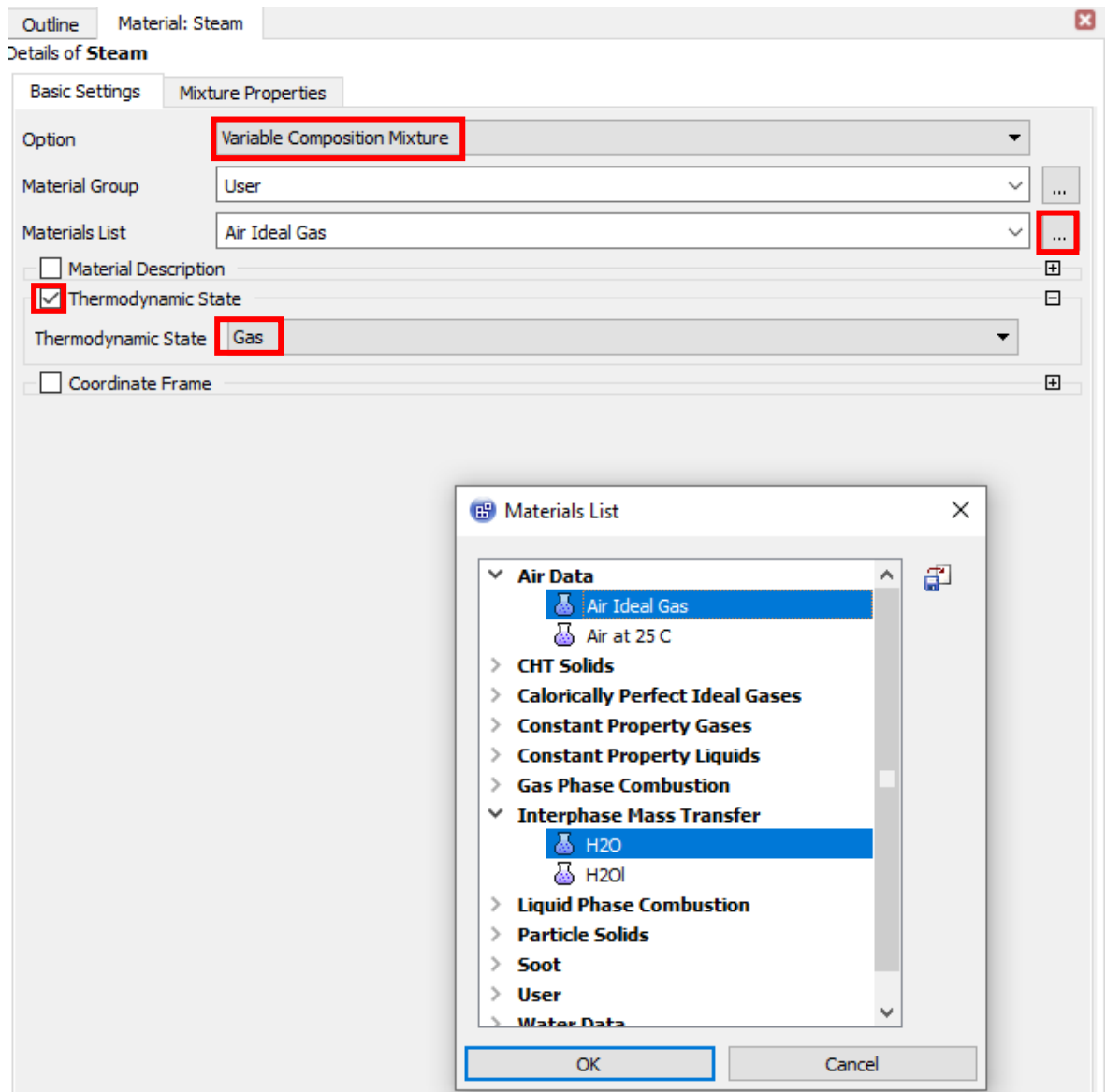


Confirm *OK*.

9) Select the icon for creating new material and name it *Steam*



10) Change *Option* into *Variable Composition Mixture* and in the field *Material List* choose *Air Ideal Gas* and *H2O*. Use the icon to choose two materials and hold down the *Ctrl* key while selecting. Submit *OK*.



Next again confirm *OK*.

Outline Material: Steam

Details of **Steam**

Basic Settings Mixture Properties

Option Variable Composition Mixture

Material Group User

Materials List Air Ideal Gas,H2O

☐ Material Description

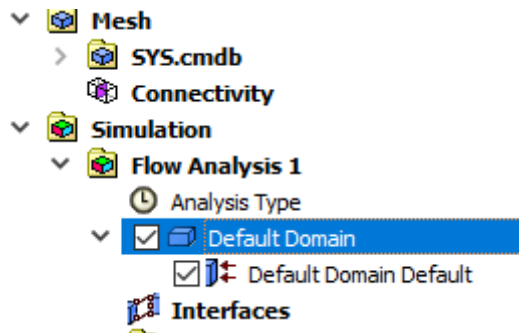
☒ Thermodynamic State

Thermodynamic State Gas

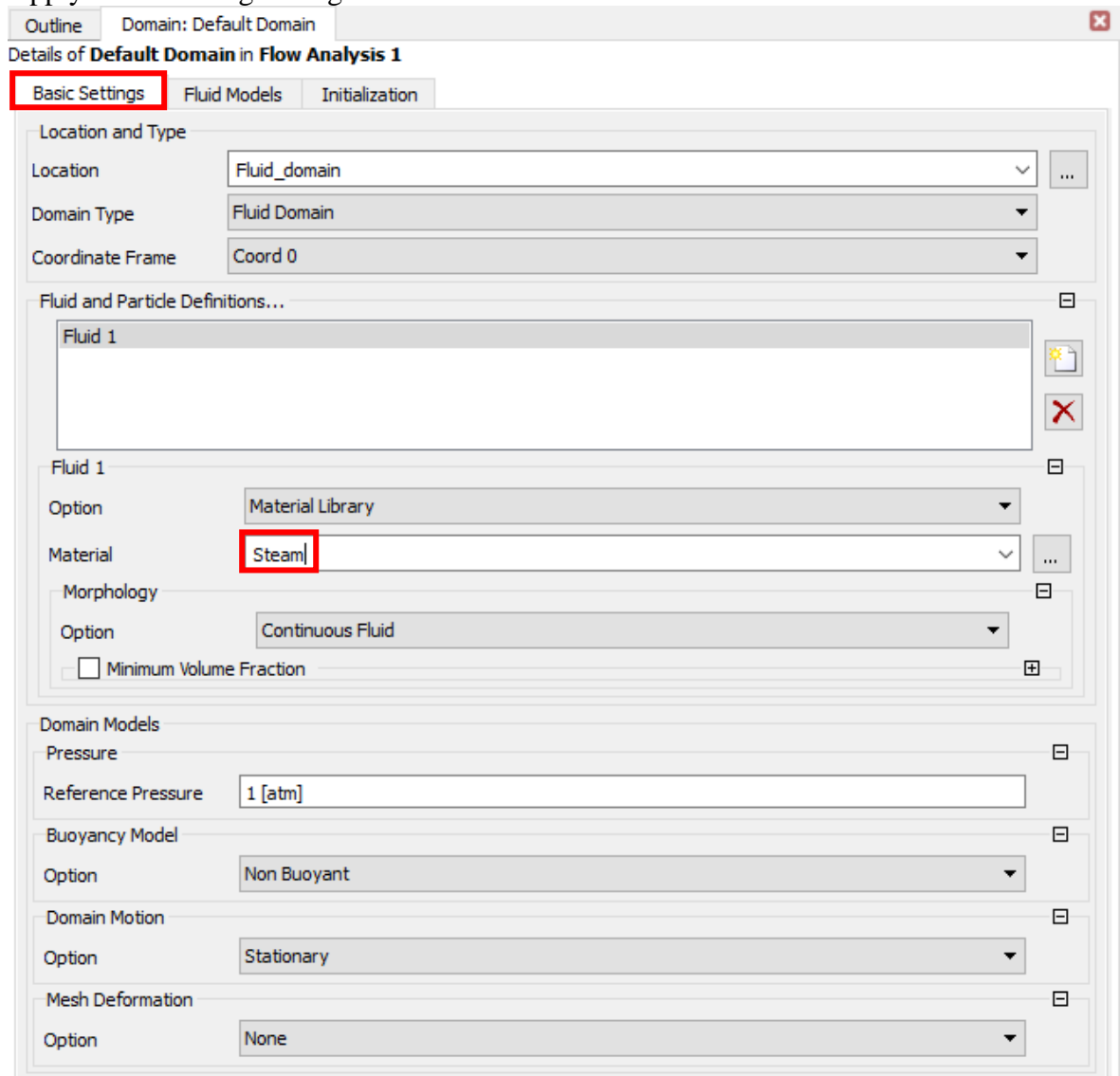
☐ Coordinate Frame

OK Apply Close

11) Double-click LMB *Default Domain*



12) Apply the following settings



Outline

Domain: Default Domain

Details of **Default Domain** in **Flow Analysis 1**

Basic Settings

Fluid Models

Initialization

Heat Transfer

Option

Thermal Energy

☐ Ind. Viscous Dissipation

Turbulence

Option

Shear Stress Transport

Wall Function

Automatic

☐ Turbulent Flux Closure for Heat Transfer

☐ Transitional Turbulence

☐ Advanced Turbulence Control

Combustion

Option

None

Thermal Radiation

Option

None

☐ Electromagnetic Model

Component Models

Wall Condensation Model

Option

Concentration Boundary Layer Model

Component

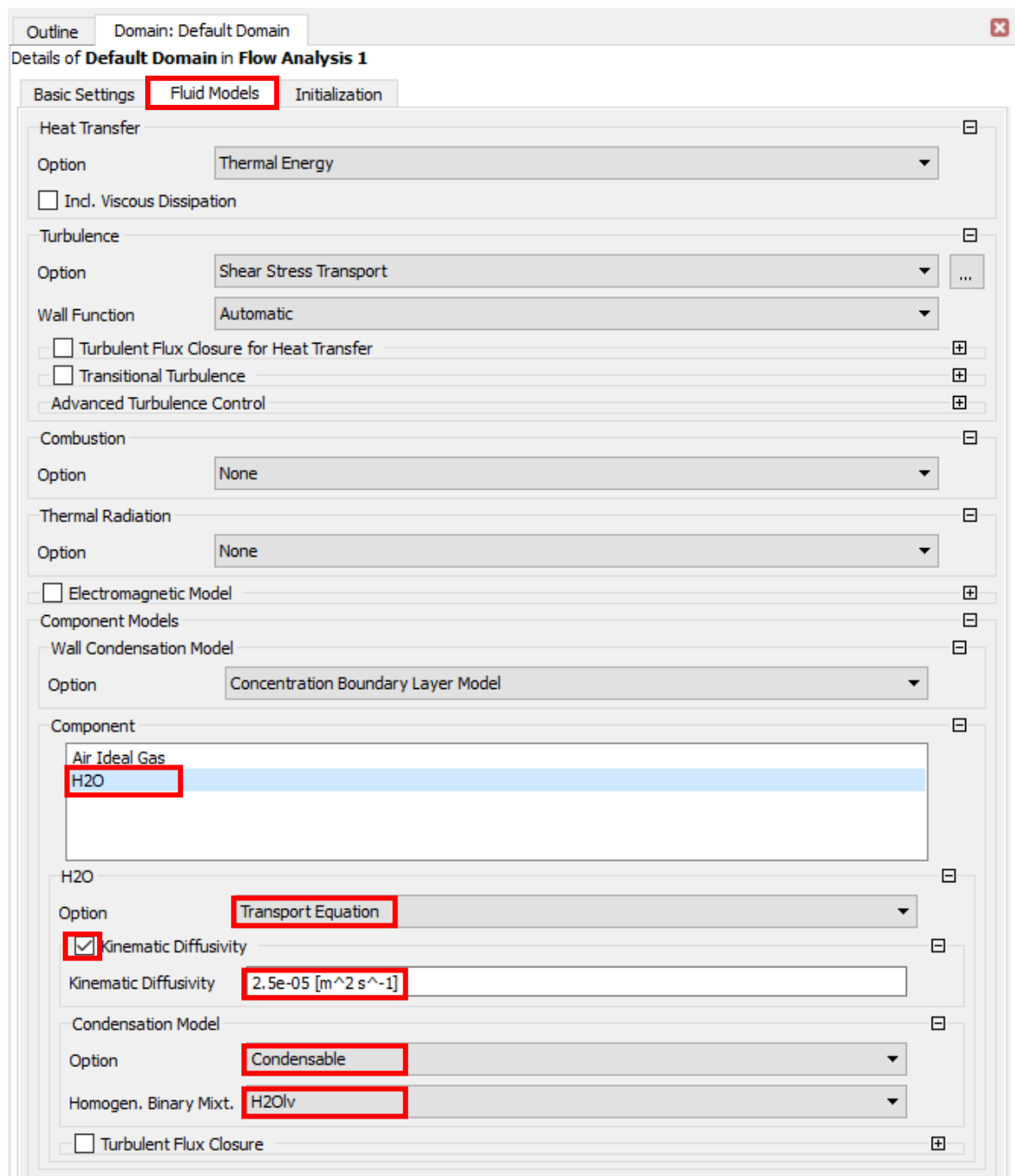
Air Ideal Gas

H2O

Air Ideal Gas

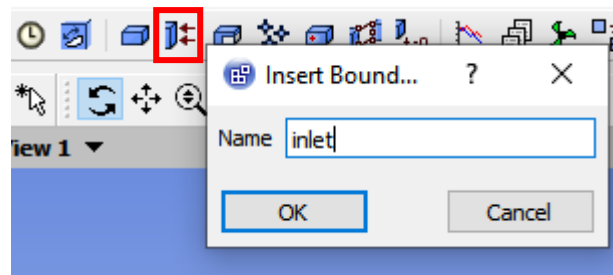
Option

Constraint

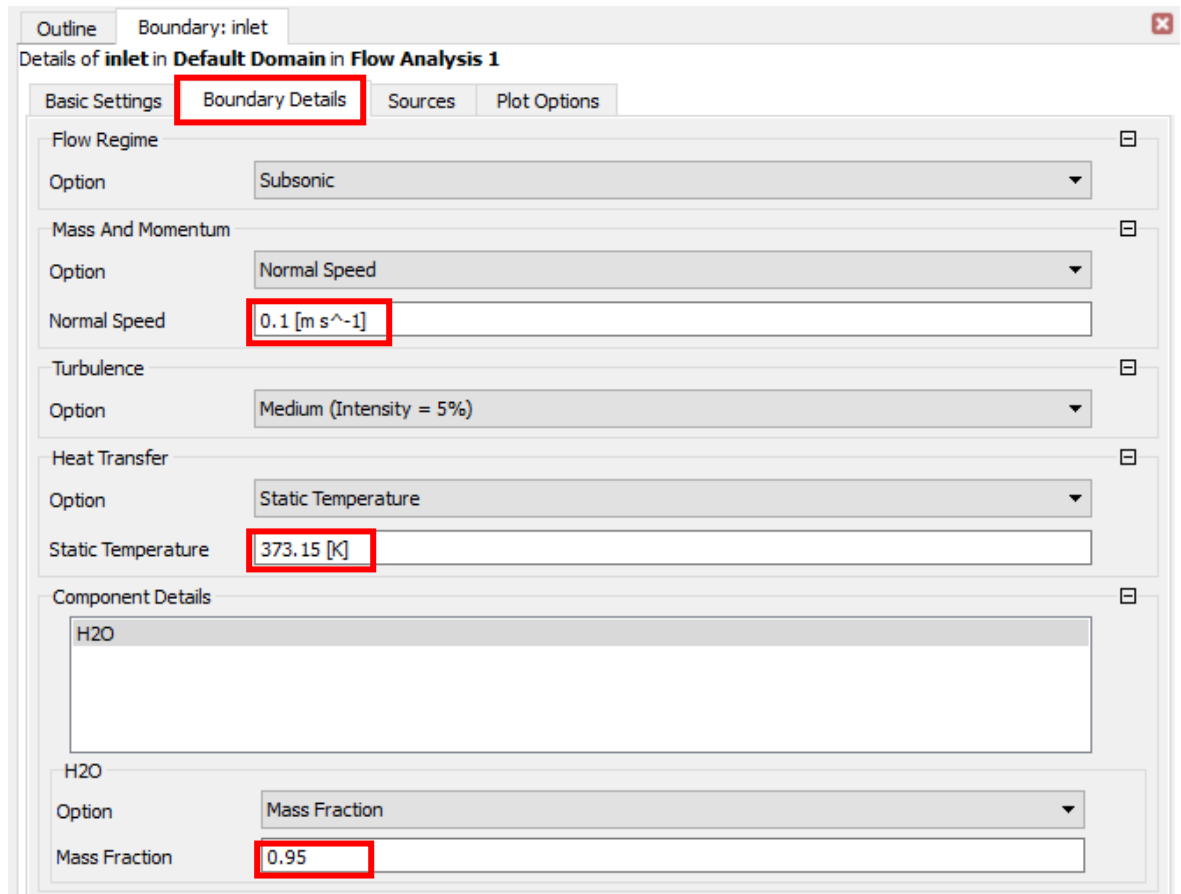
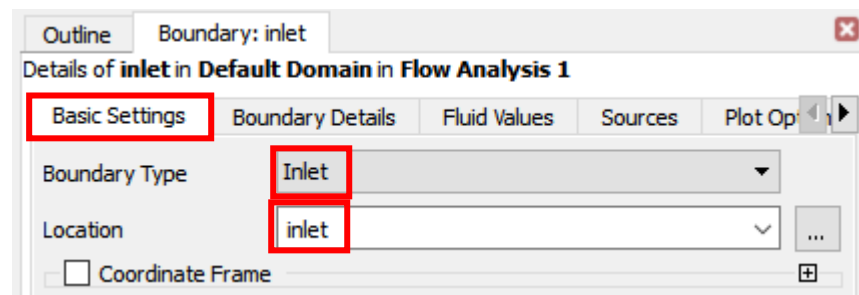


Confirm *OK*.

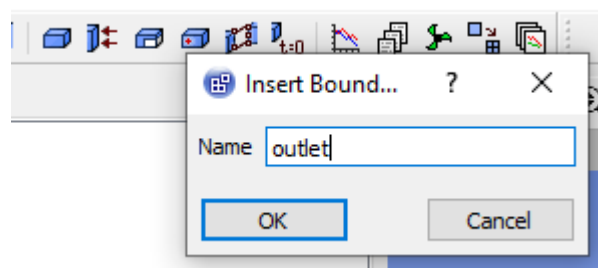
13) Create an boundary condition named *inlet*

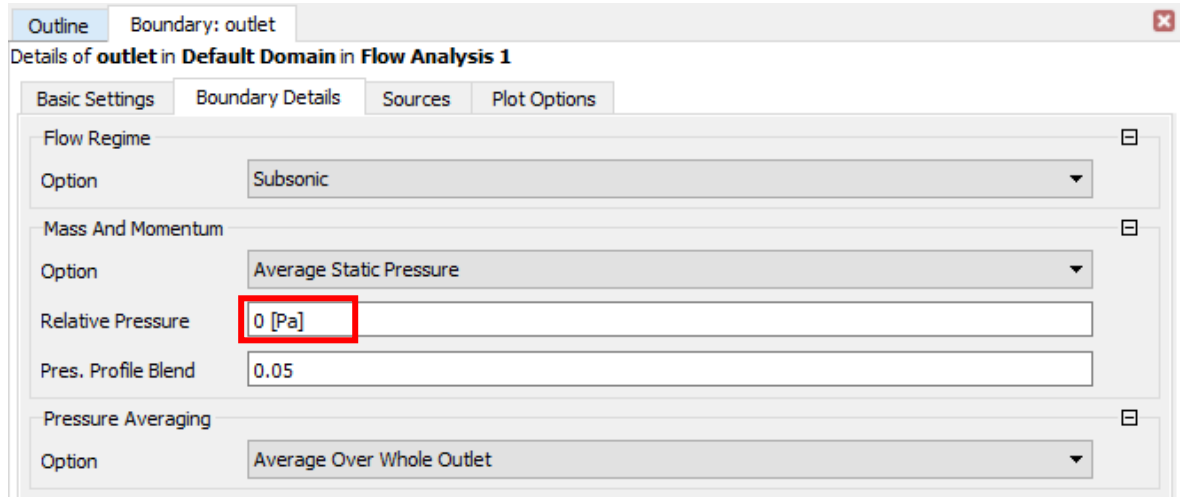
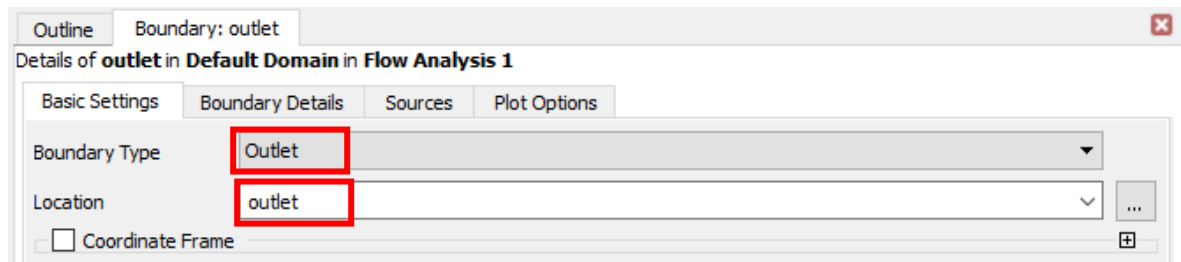


Apply the settings as below and confirm *OK*.

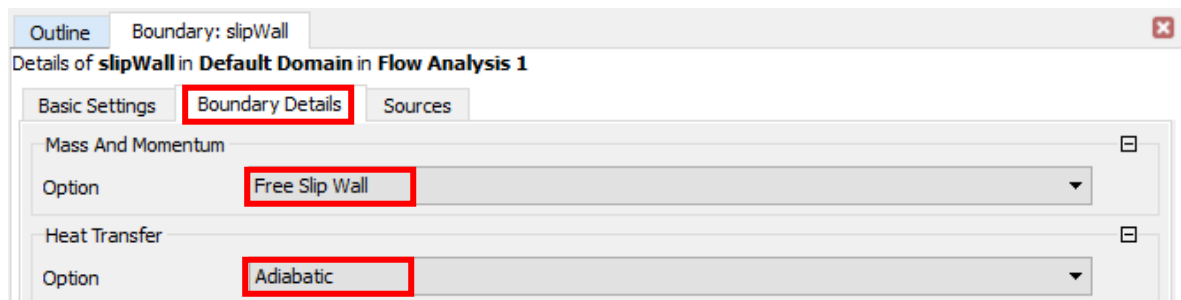
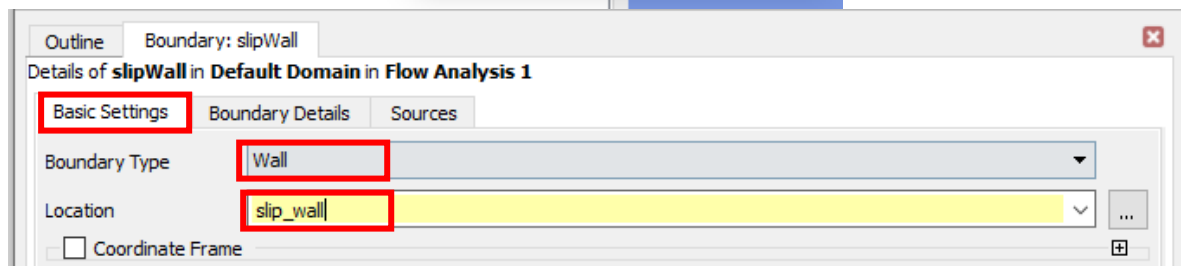
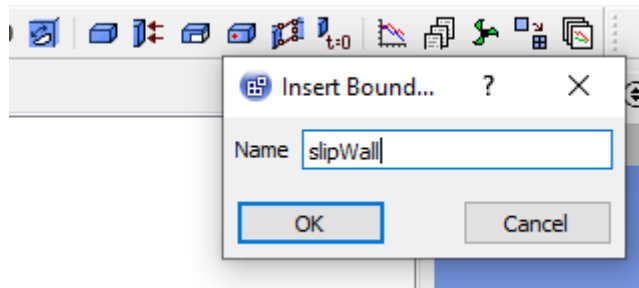


14) Create an boundary condition *outlet*

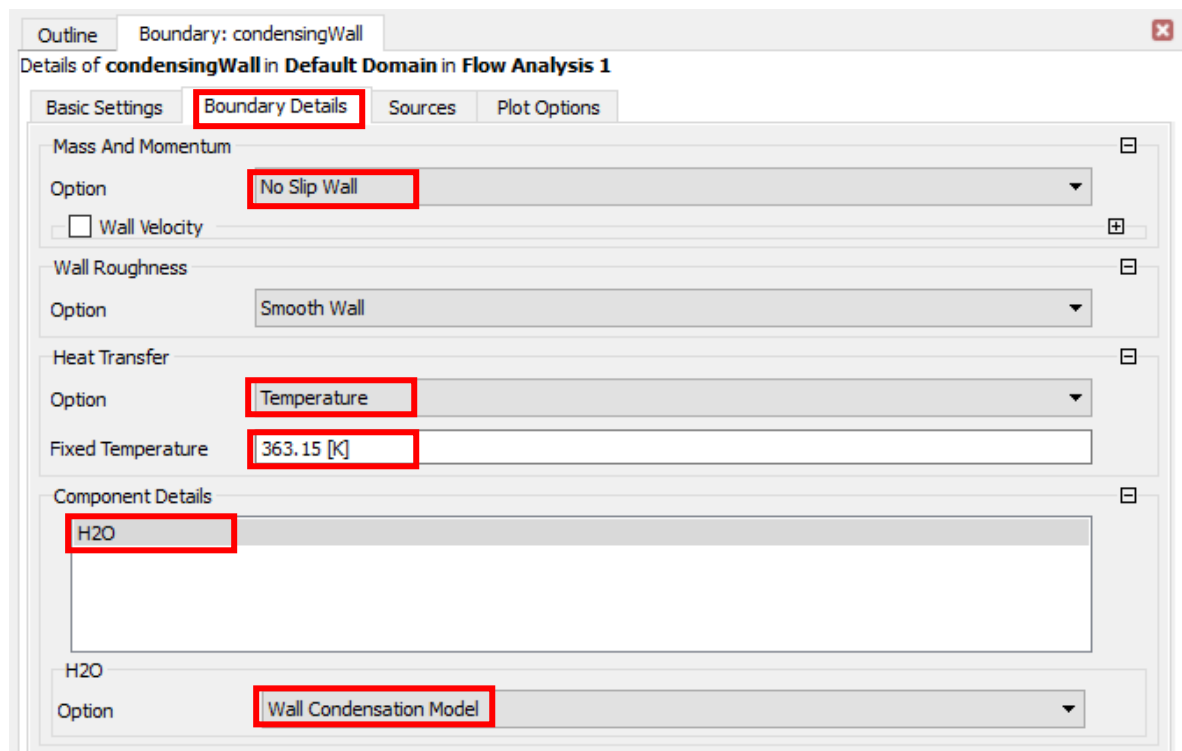
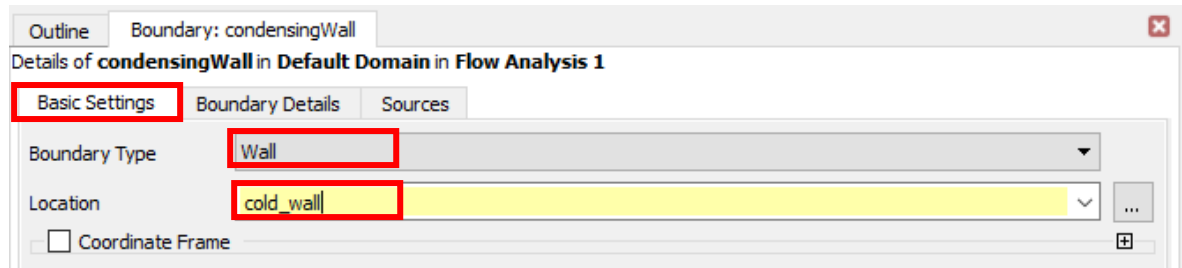
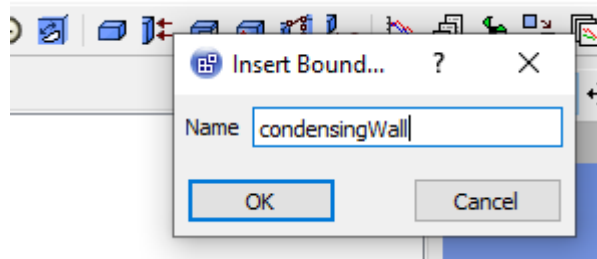




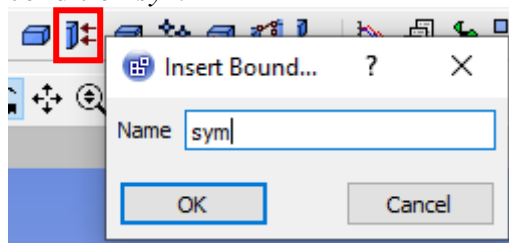
15) Create an boundary condition *slipWall*



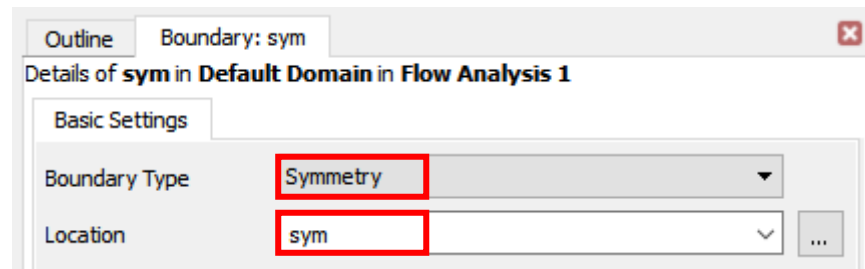
16) Create an boundary condition *condensingWall*



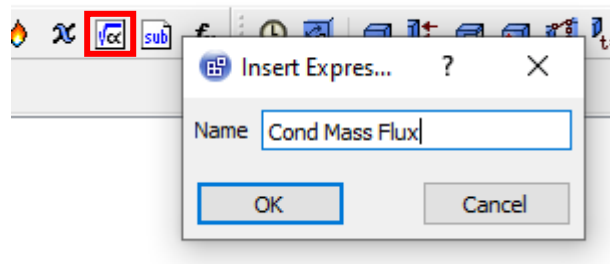
17) Create an boundary condition *sym*



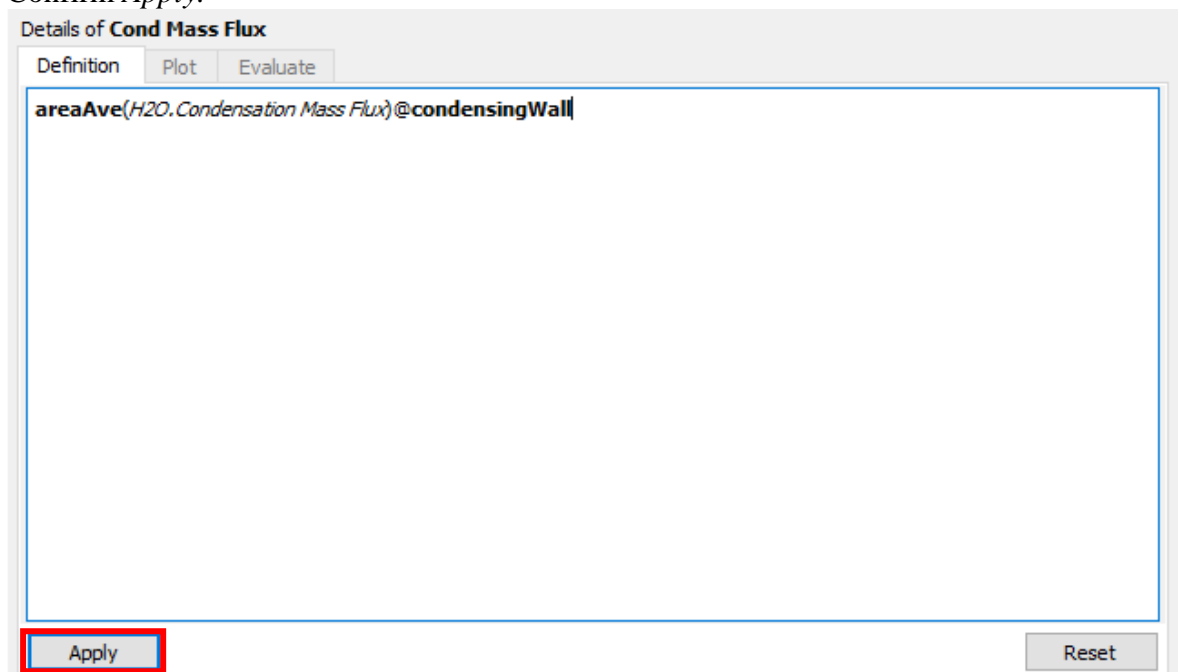
Apply the settings as below and confirm *OK*.



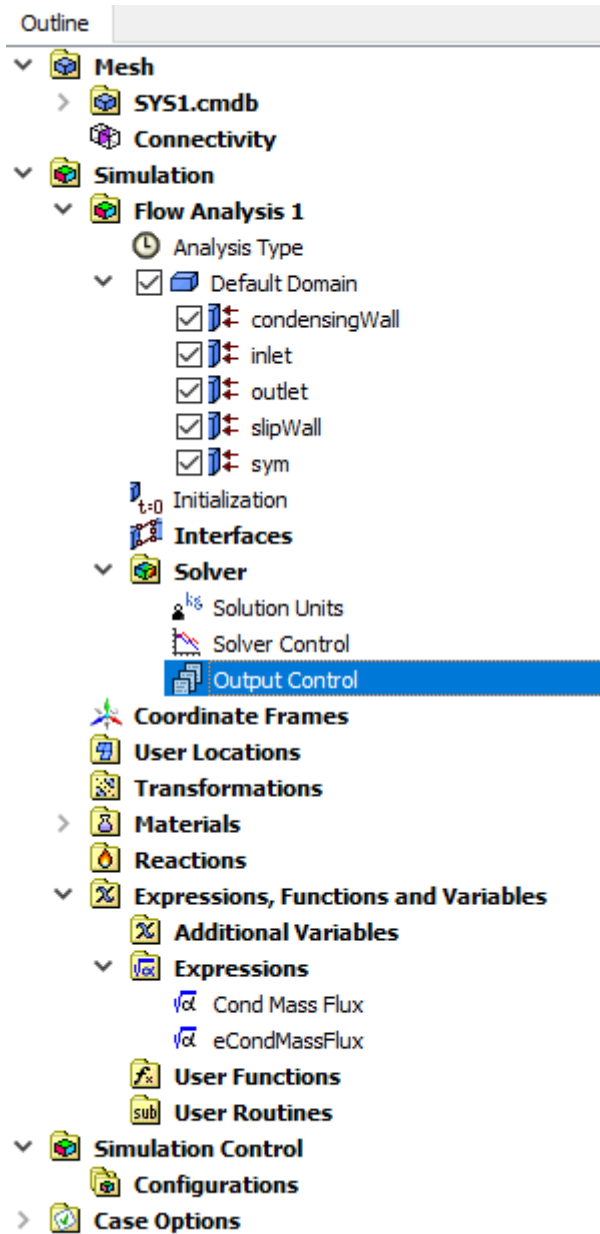
18) Create *expression* named *Cond Mass Flux*



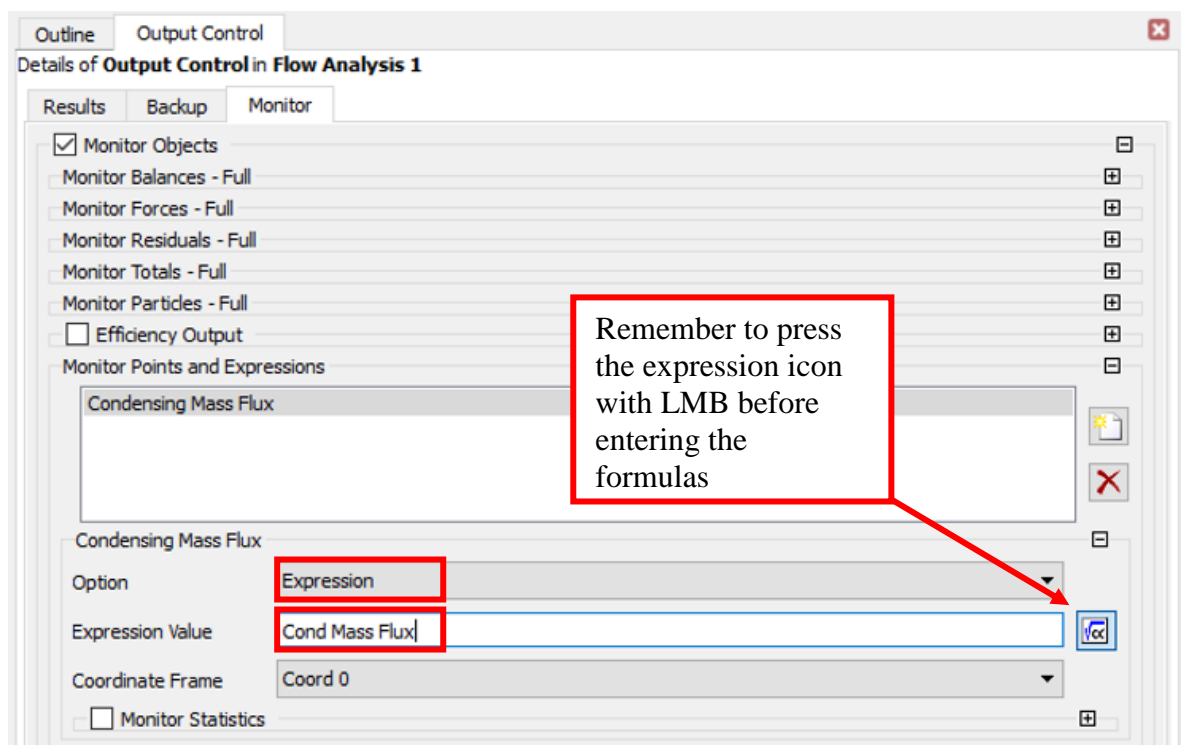
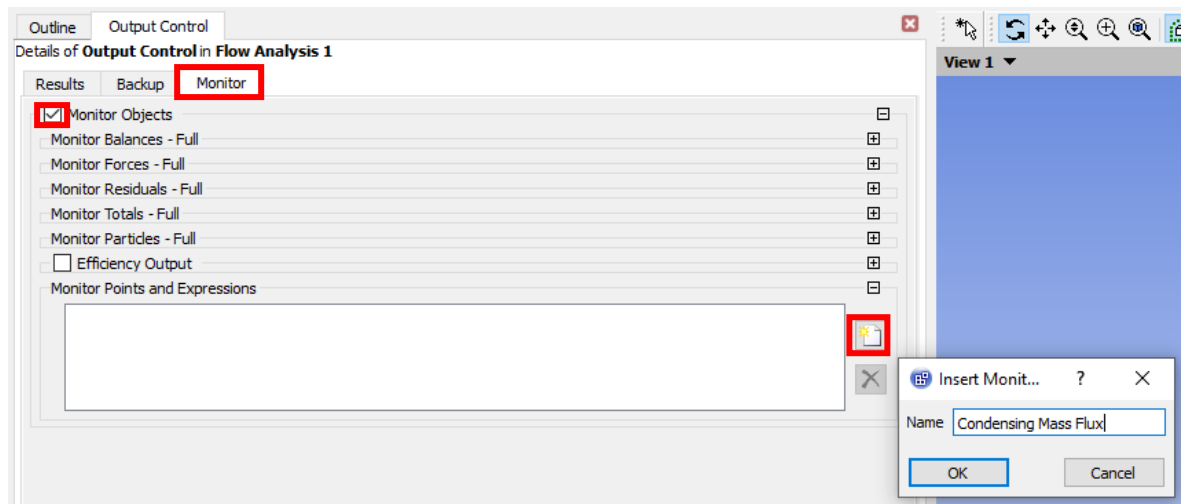
19) Apply the following definition: `areaAve(H2O.Condensation Mass Flux)@condensingWall`
Confirm *Apply*.



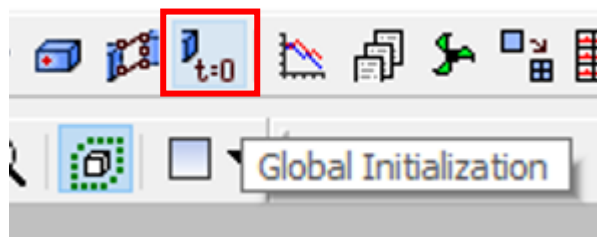
20) Open *Output Control*



Apply the following settings and confirm *OK*.



- 21) Create initial conditions by clicking the LMB icon *Global Initialization* (at the top of the screen, almost in the middle)



Apply the settings as below and confirm *OK*.

Outline Initialization ✕

Details of **Global Initialization** in **Flow Analysis 1**

Global Settings

☐ Coordinate Frame ⊕

Initial Conditions ⊖

Velocity Type Cartesian ⌵

Cartesian Velocity Components ⊖

Option Automatic with Value ⌵

U 0.1 [m s⁻¹]

V 0 [m s⁻¹]

W 0 [m s⁻¹]

Static Pressure ⊖

Option Automatic with Value ⌵

Relative Pressure 0 [Pa]

Temperature ⊖

Option Automatic with Value ⌵

Temperature 373.15 [K]

Component Details ⊖

H2O
































H2O

Option Automatic with Value ⌵

Mass Fraction 0

22) OpenSolver Control i apply below settings

Outline

- ✓  **Mesh**
 - >  **SYS1.cmdb**
 -  **Connectivity**
- ✓  **Simulation**
 - ✓  **Flow Analysis 1**
 -  Analysis Type
 - ✓  **Default Domain**
 - ☒  **condensingWall**
 - ☒  **inlet**
 - ☒  **outlet**
 - ☒  **slipWall**
 - ☒  **sym**
 -  Initialization
 -  **Interfaces**
 - ✓  **Solver**
 -  Solution Units
 -  **Solver Control**
 -  Output Control
 -  **Coordinate Frames**
 -  **User Locations**
 -  **Transformations**
 - >  **Materials**
 -  **Reactions**
 - ✓  **Expressions, Functions and Variables**
 -  **Additional Variables**
 -  **Expressions**
 -  **User Functions**
 -  **User Routines**
 - ✓  **Simulation Control**
 -  **Configurations**
 - >  **Case Options**

The screenshot shows the 'Solver Control' dialog box for 'Flow Analysis 1'. The 'Basic Settings' tab is active. The 'Advection Scheme' is set to 'High Resolution'. The 'Turbulence Numerics' are also set to 'High Resolution'. Under 'Convergence Control', the 'Min. Iterations' and 'Max. Iterations' are both set to 600. The 'Fluid Timescale Control' section shows 'Timescale Control' set to 'Auto Timescale', 'Length Scale Option' set to 'Conservative', and 'Timescale Factor' set to 1.0. The 'Convergence Criteria' section shows 'Residual Type' set to 'RMS' and 'Residual Target' set to 1e-6. There are checkboxes for 'Conservation Target', 'Elapsed Wall Clock Time Control', and 'Interrupt Control', all of which are currently unchecked.

Outline Solver Control

Details of **Solver Control** in **Flow Analysis 1**

Basic Settings Equation Class Settings Advanced Options

Advection Scheme

Option High Resolution

Turbulence Numerics

Option High Resolution

Convergence Control

Min. Iterations 600

Max. Iterations 600

Fluid Timescale Control

Timescale Control Auto Timescale

Length Scale Option Conservative

Timescale Factor 1.0

☐ Maximum Timescale

Convergence Criteria

Residual Type RMS

Residual Target 1e-6

☐ Conservation Target

☐ Elapsed Wall Clock Time Control

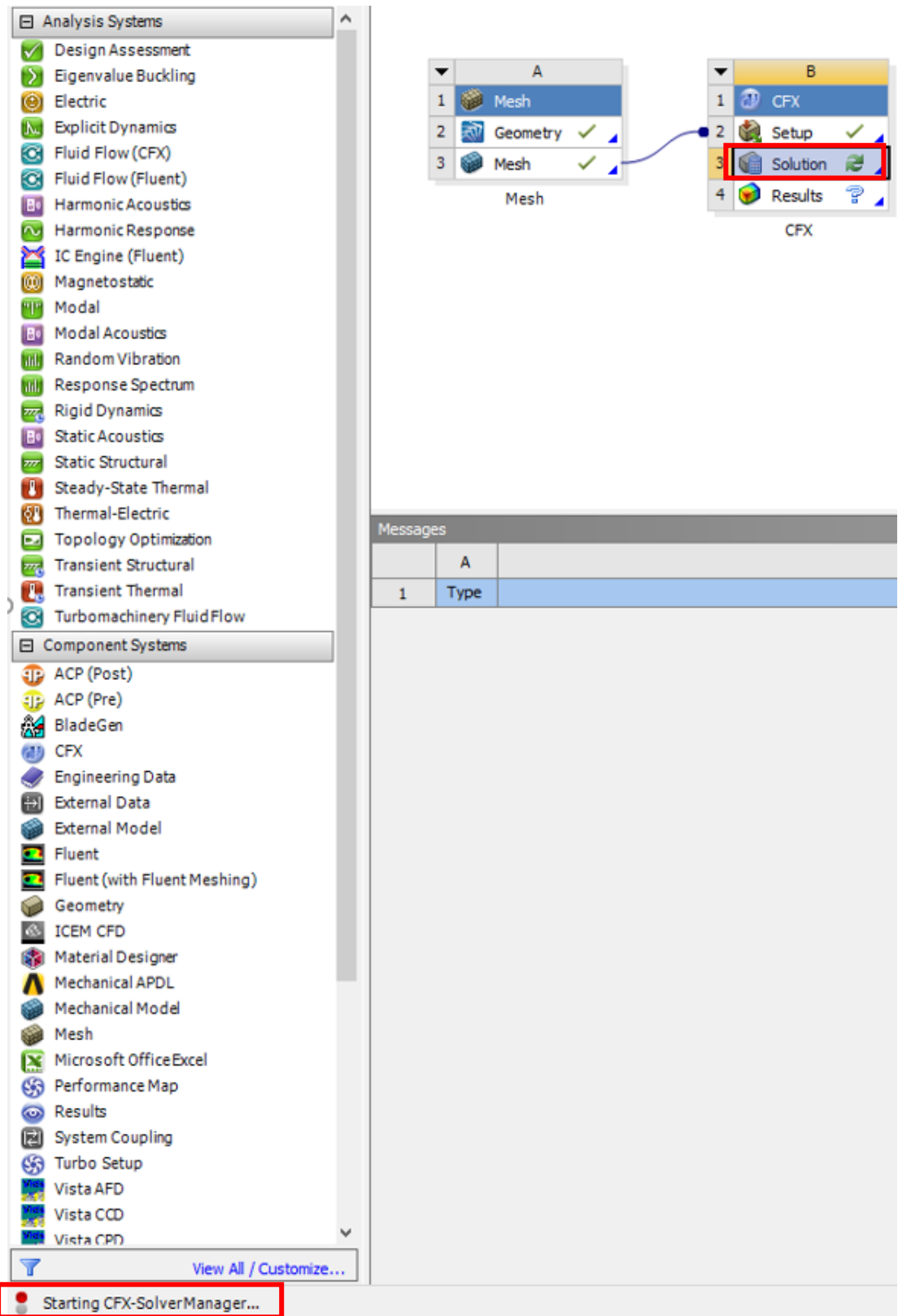
☐ Interrupt Control

Confirm *OK*.

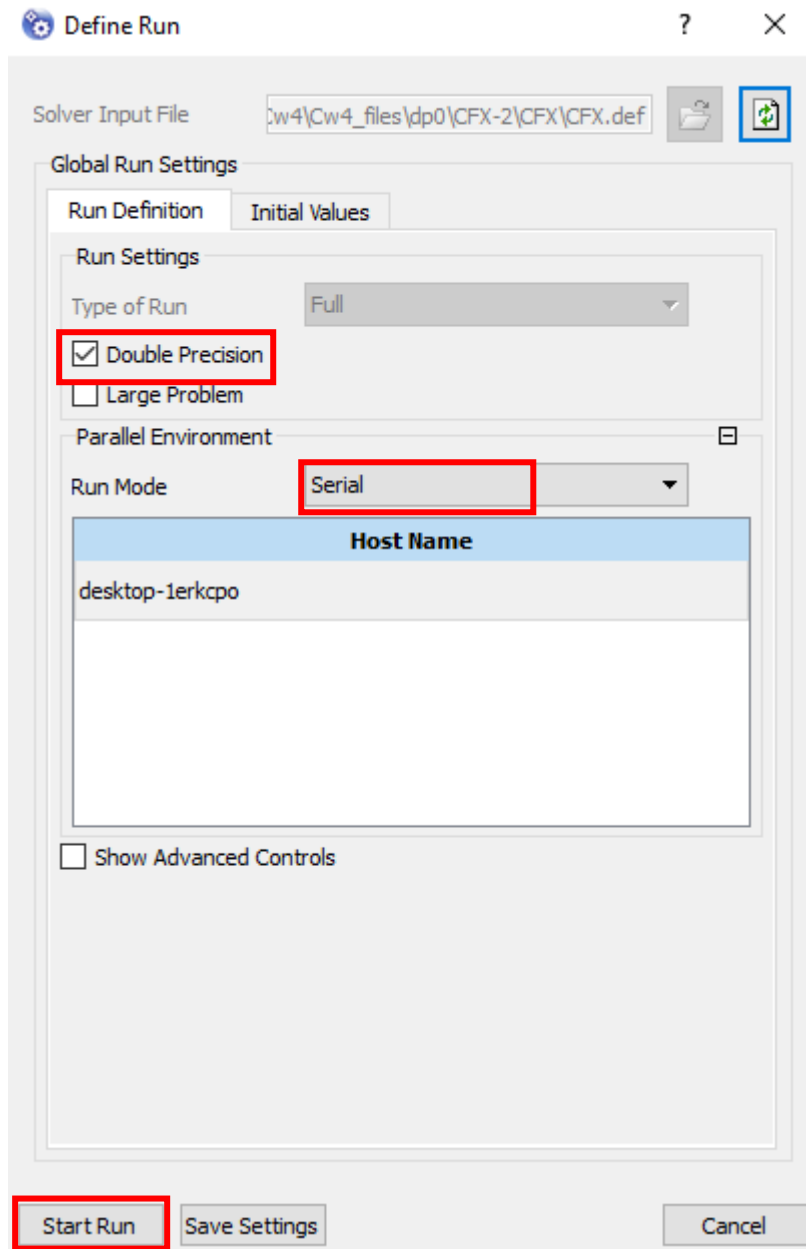
23) Close *Ansys CFX*.

2.4. CALCULATIONS

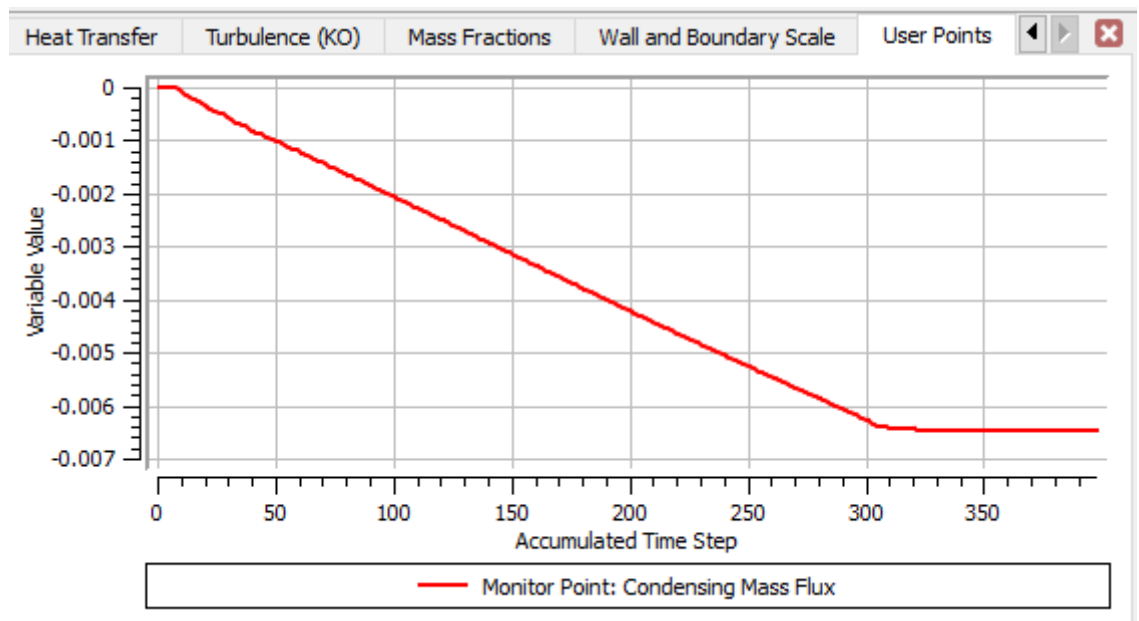
- 1) Double-click on *Solution* to run *Ansys CFX Solver Manager*



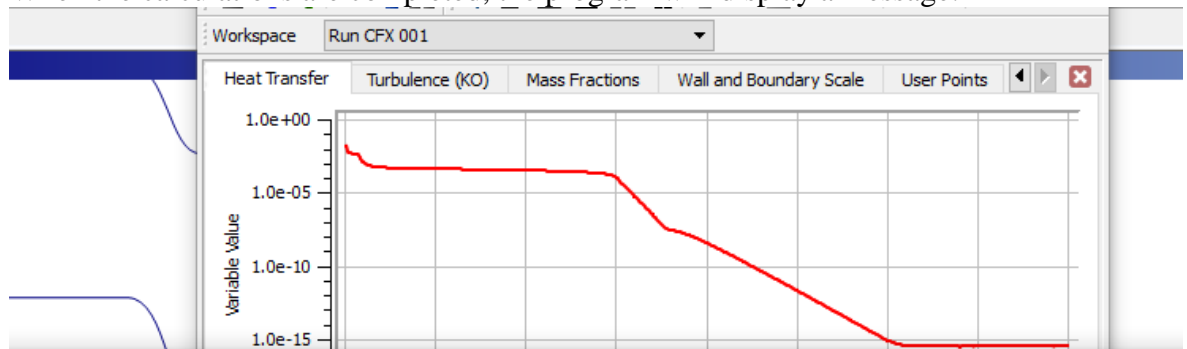
- 2) Apply below settings and press *Start Run*. The program will perform calculations. Wait a few moments for the message about completing the calculations.



- 3) Calculations take about 10 minutes. Watch the individual bookmark tabs as they change. Pay particular attention to the *User Point* tab, where the stream of condensed steam is shown in each iteration. Steady state will be reached when the curve stabilizes, which will occur after about 350 iterations. An additional 250 iterations (600 iterations were set up in point 2.3.22) are carried out so that all residuals also stabilize.



4) When the calculations are completed, the program will display a message:



Solver Run Finished Normally

CFX_001 has completed normally.
Run concluded at: śr. 8. kwi 12:33:29 2020
Results are in
I:/KompPraca_2020III13/DYDAKTYKA/PRZEDMIOTY/WZPCP/D/Cw4/Cw4_pending/dp0_CFX_3_Solution_3/CFX_001.res

[open this workspace now](#)

OK

End of solution stage.

```

+-----+
| The results from this run of the ANSYS CFX Solver have been |
| written to |
| I:/KompPraca_2020III13/DYDAKTYKA/PRZEDMIOTY/WZPCP/D/Cw4/Cw4_pending/dp0_CFX_3_Solution_3/CFX_001.res |
+-----+

+-----+
| For CFX runs launched from Workbench, the final locations of |
| directories and files generated may differ from those shown. |
+-----+

This run of the ANSYS CFX Solver has finished.

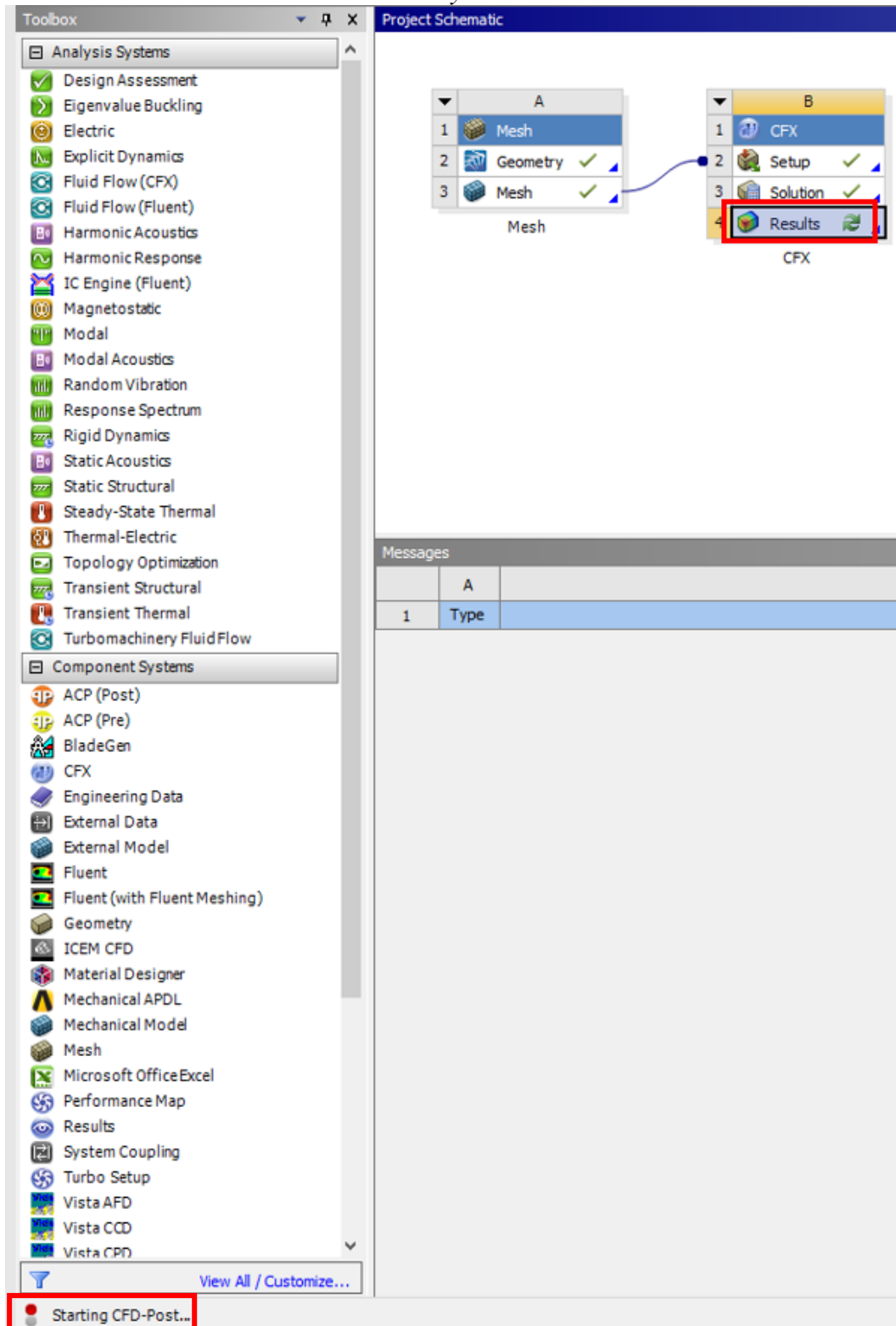
```

Run Complete

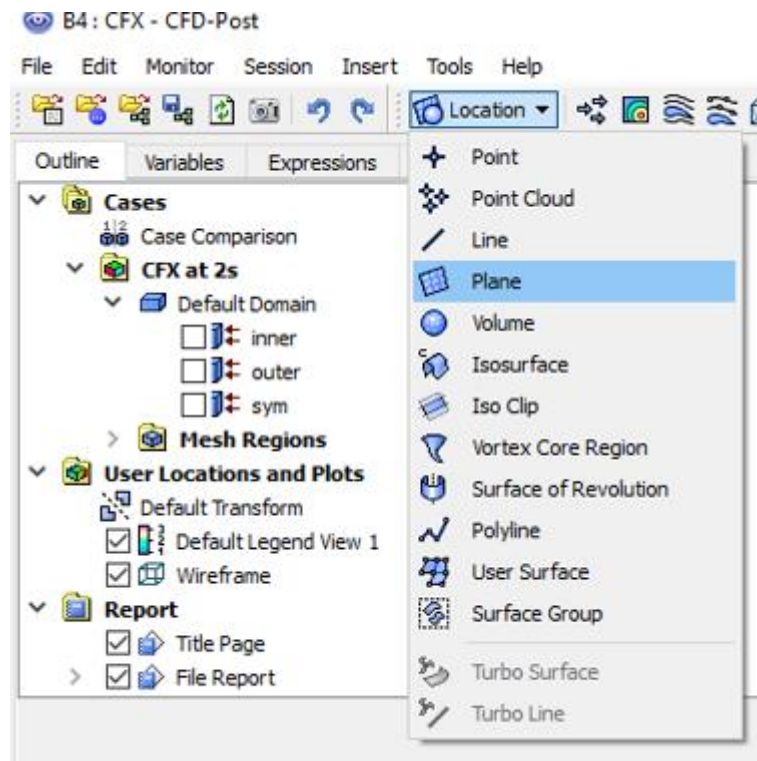
5) Confirm OK and close *Ansys CFX Solver Manager*. Save project in *Workbench*.

2.5. ELABORATION OF THE RESULTS

- 1) Double-click LMB on *Results* to run *Ansys CFD Post* and see the results



- 2) From menu *Location* choose *Plane*

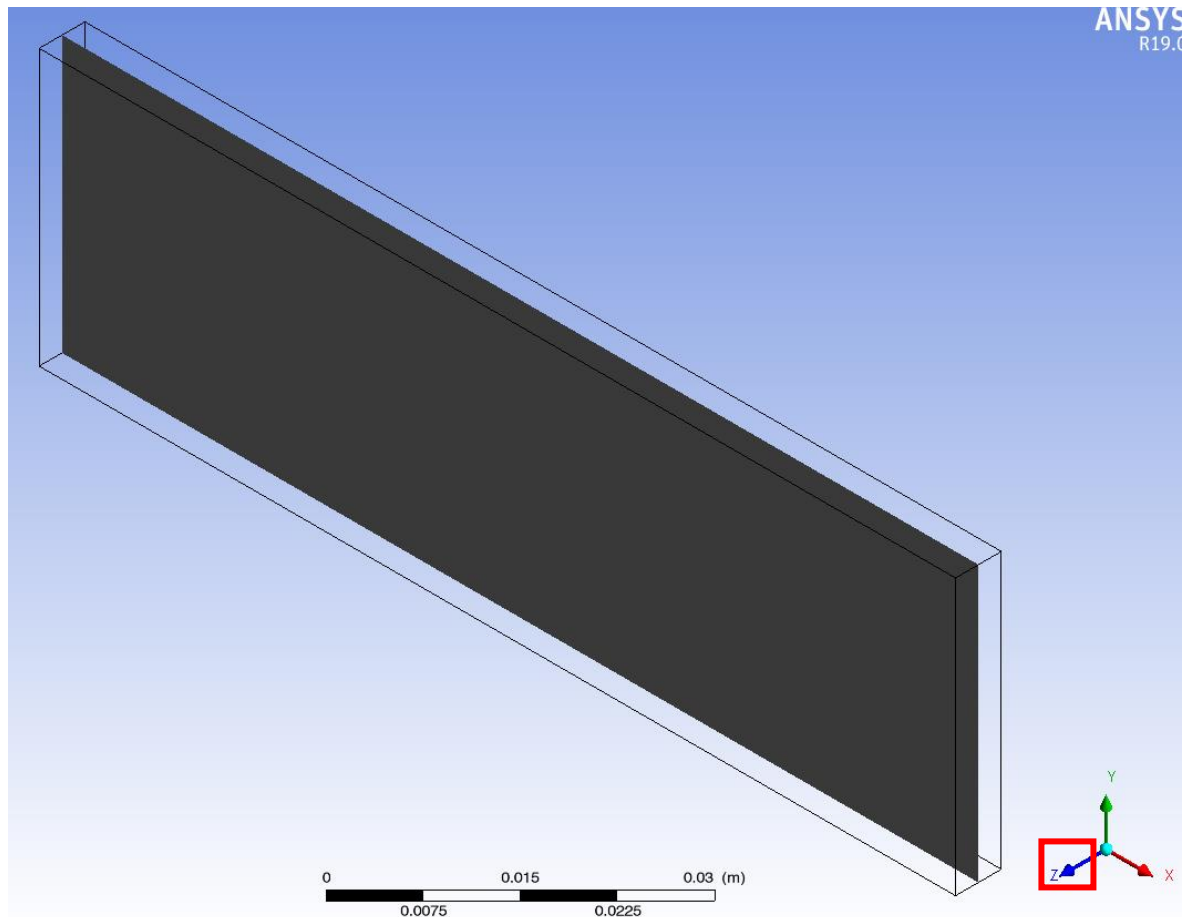


Apply below settings and confirm *Apply*.

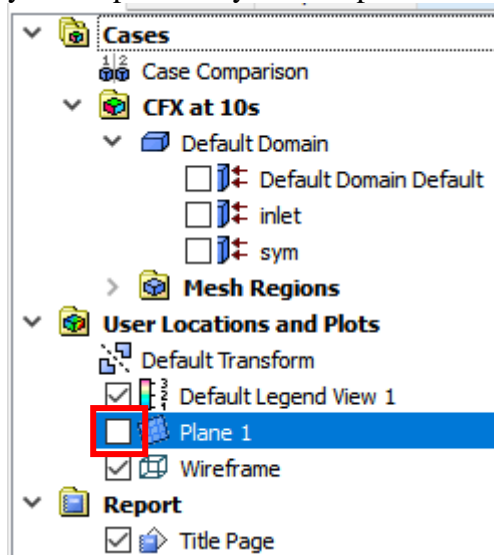
Details of **Plane 1**

Geometry	Color	Render	View
Domains	All Domains		
Definition	<div>Method</div> <div>XY Plane</div> <div>Z</div> <div>0.0025 [m]</div>		
Plane Bounds	<div>Type</div> <div>None</div>		
Plane Type	<div> <input checked="" type="radio"/> Slice <input type="radio"/> Sample </div>		

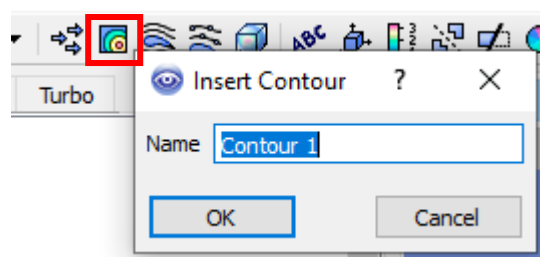
3) LMB press X axis



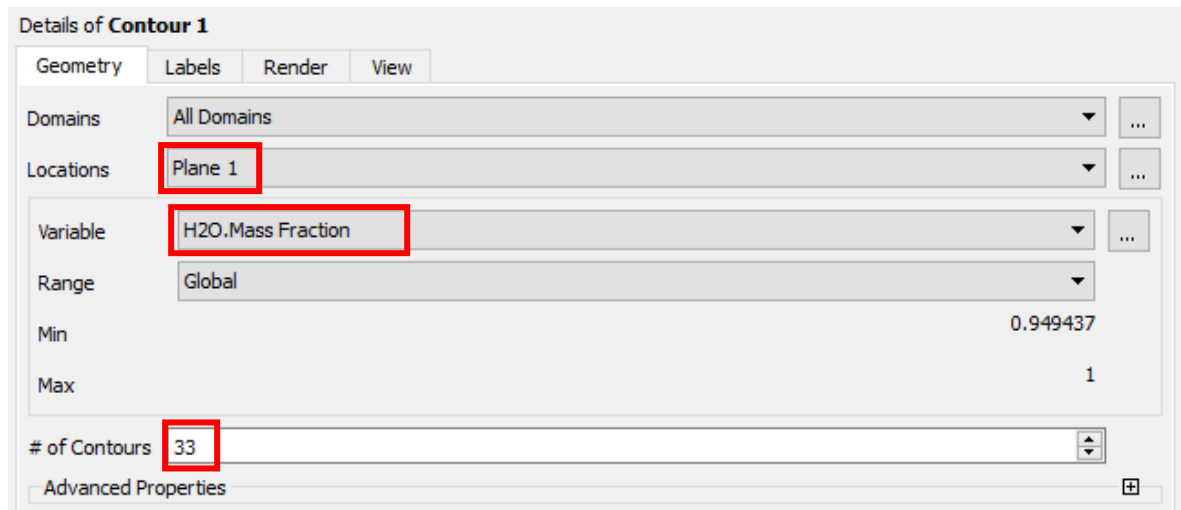
4) Uncheck the visibility of the previously created plane.



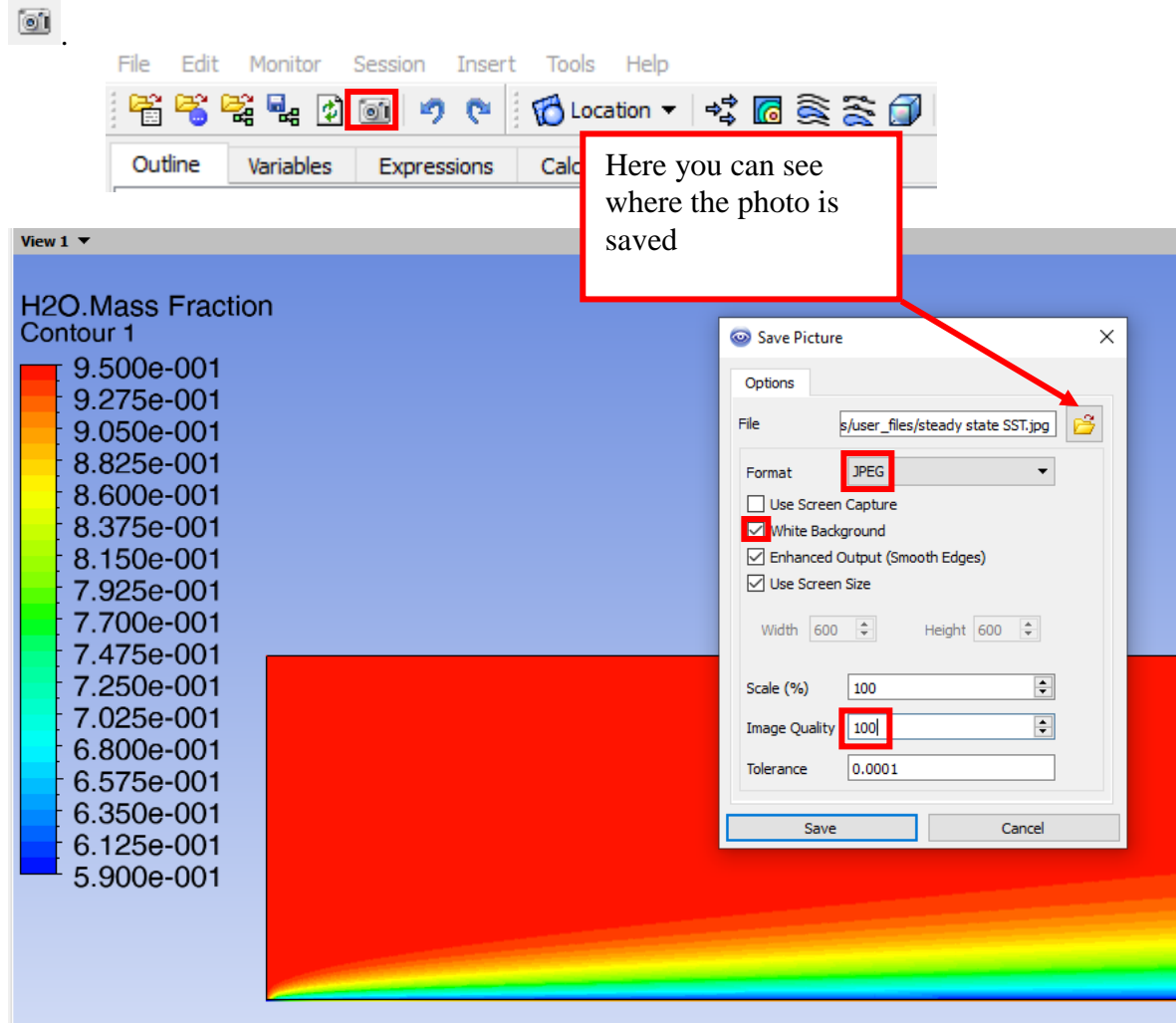
5) Select contour creation and confirm OK.



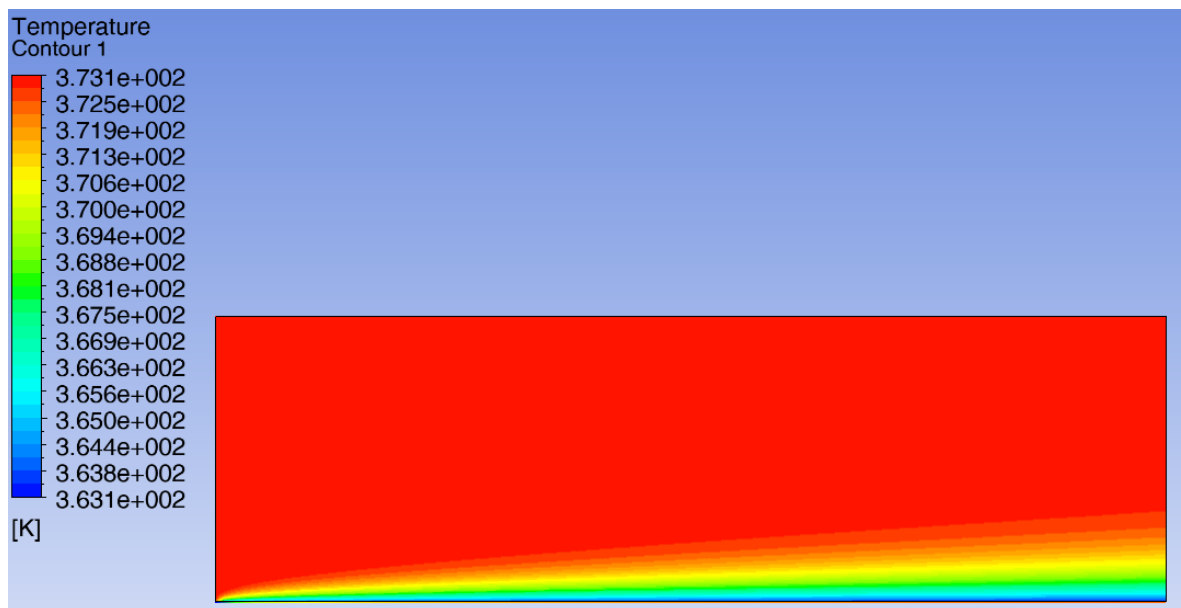
Apply below settings and confirm *Apply*.



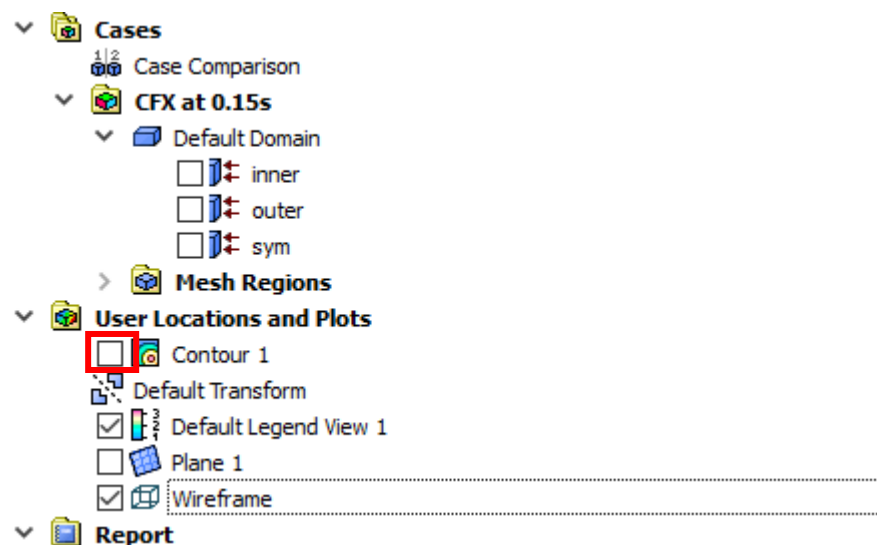
- 6) Save the photo of the water fraction contours. You can also use for this the icon



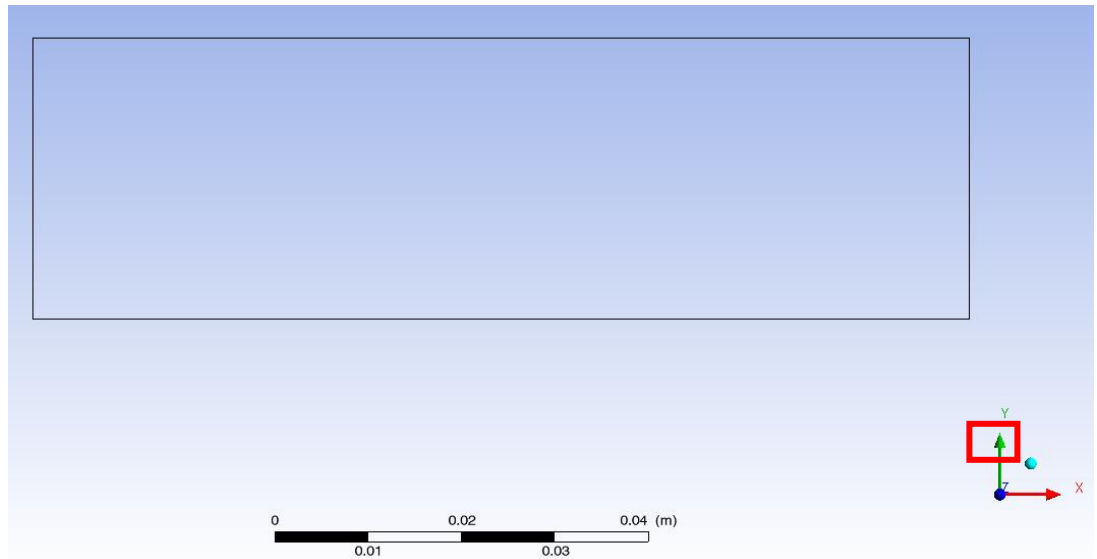
- 7) Save the photo of the temperature contours. To do this, double-click LMB on the contours created in the tree and change the variable to *Temperature*.




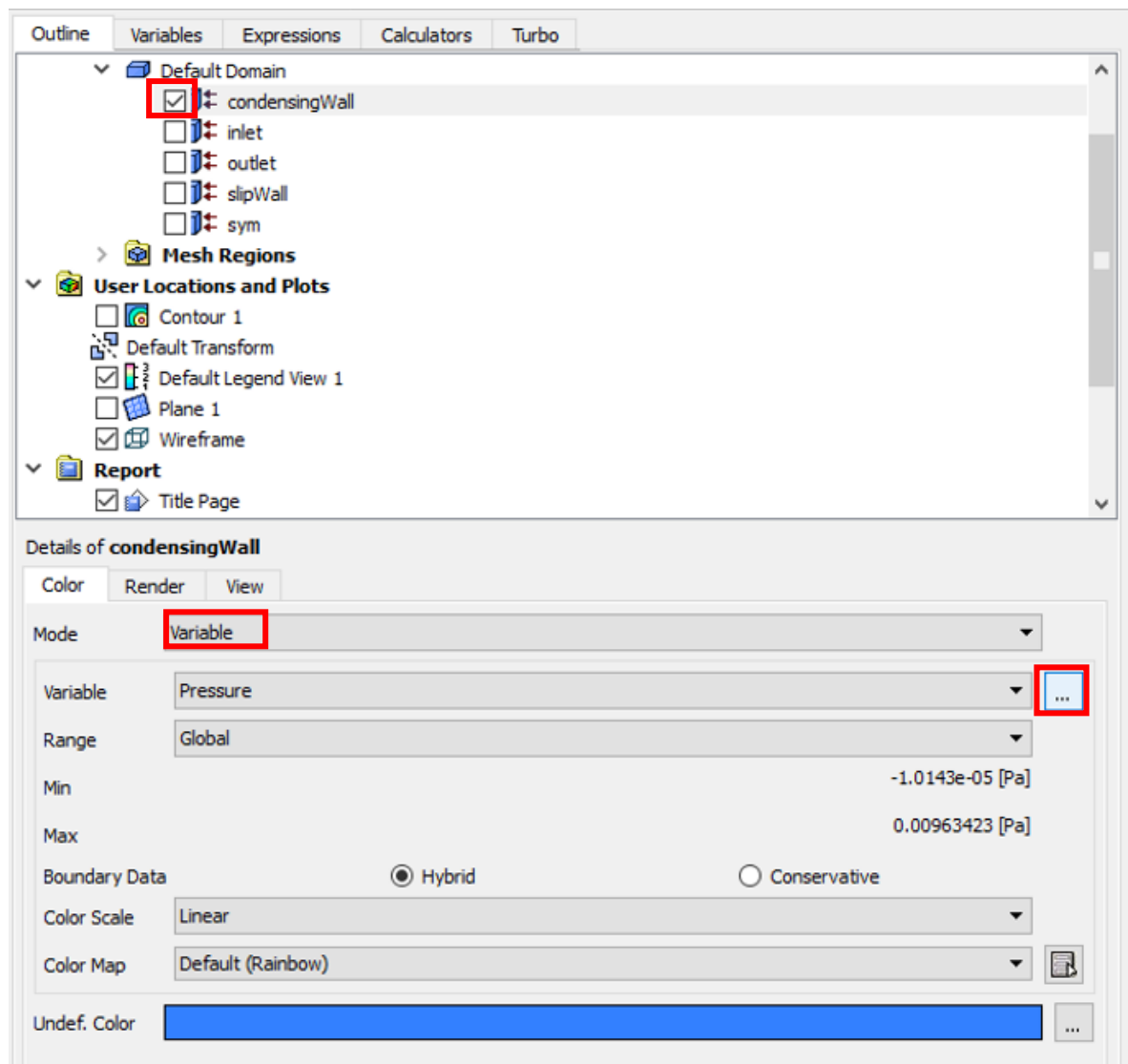
- 8) Save the photo of the speed contours. To do this, double-click LMB on the contours created in the tree and change the variable to *Velocity*.
- 9) Then turn off contour visibility.



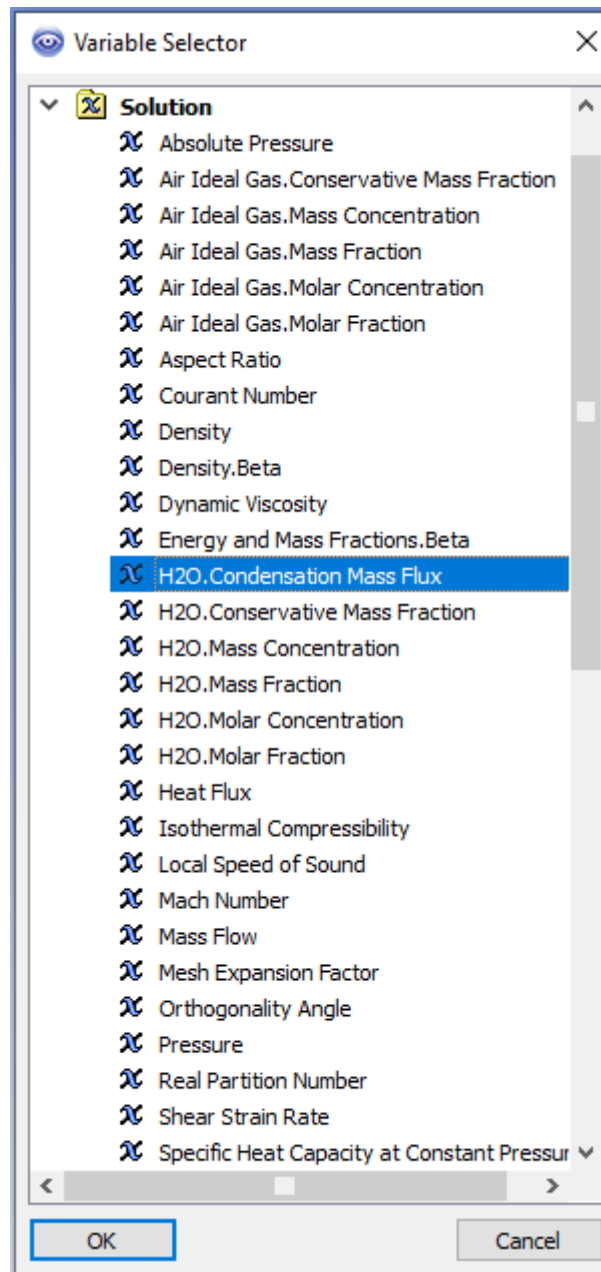
10) Click LMB on Y axis



11) Turn on *condensingWall* surface visibility in the system tree and next double-click LMB on name *condensingWall*. apply below settings and click  icon



Select *H2O.Condensation Mass Flux* variable and confirm *OK*.



Confirm *Apply*.

Details of **condensingWall**

Color Render View

Mode Variable

Variable H2O,Condensation Mass Flux

Range Global

Min -9.95362e-07 [kg s⁻¹ m⁻²]

Max 0 [kg s⁻¹ m⁻²]

Boundary Data ☒ Hybrid ☐ Conservative

Color Scale Linear

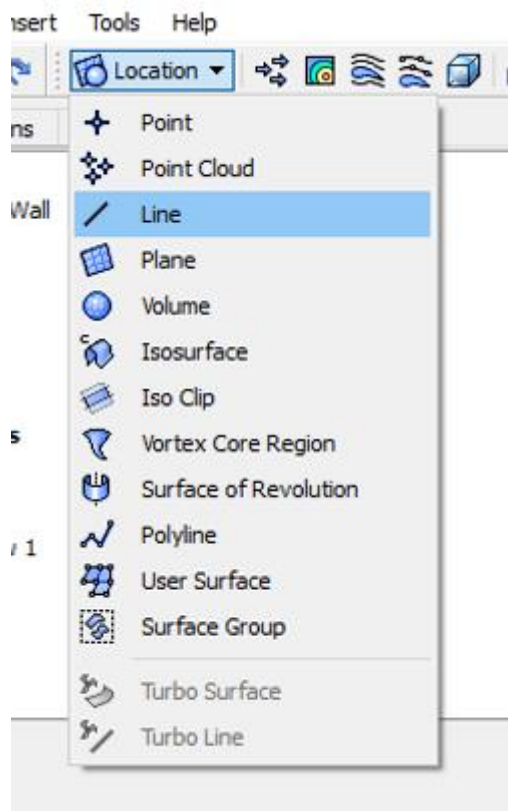
Color Map Default (Rainbow)

Undef. Color

Apply Reset Defaults

Save the photo.

- 12) Turn off visibility of the *condensingWall* surface in the system tree.
- 13) Create a line and apply the following settings:



Details of **Line 1**

Geometry Color Render View

Domains All Domains ...

Definition

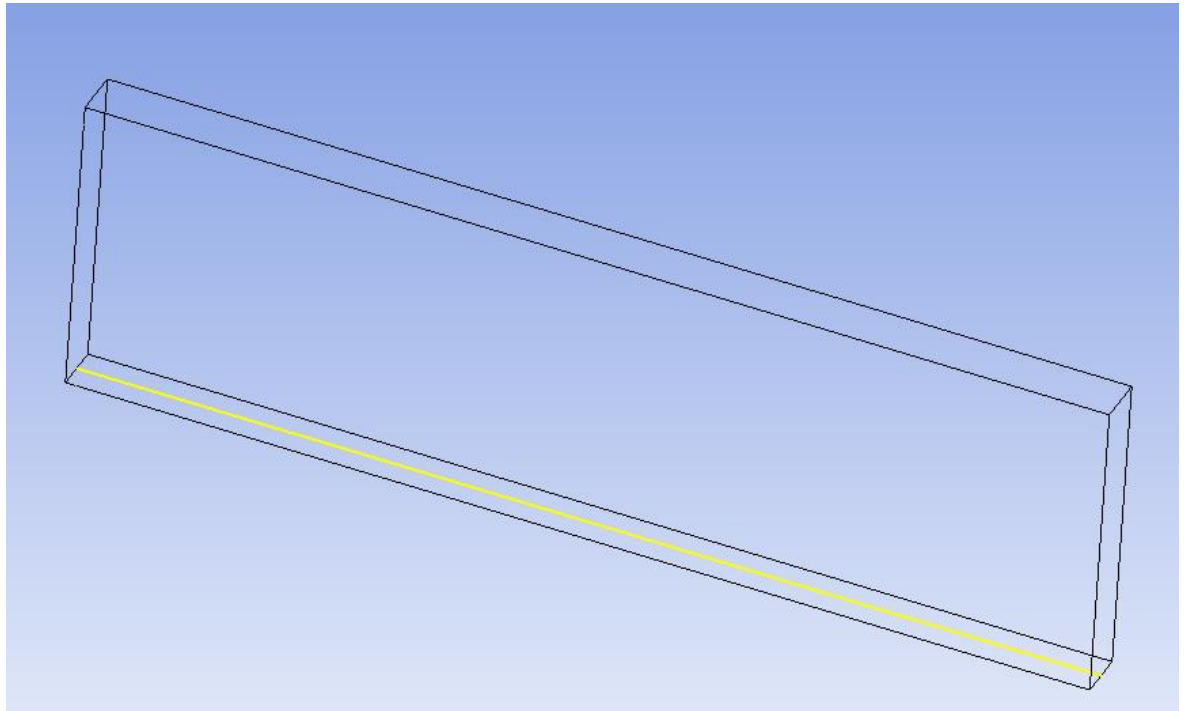
Method Two Points

Point 1	0	0	0.0025
Point 2	0.1	0	0.0025

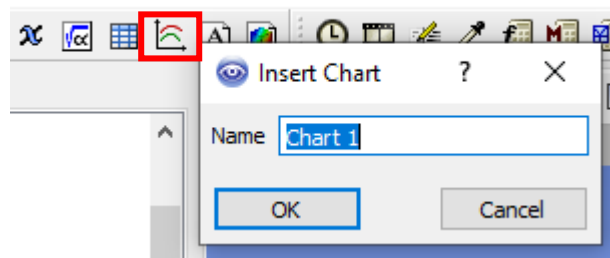
Line Type

☐ Cut ☒ Sample

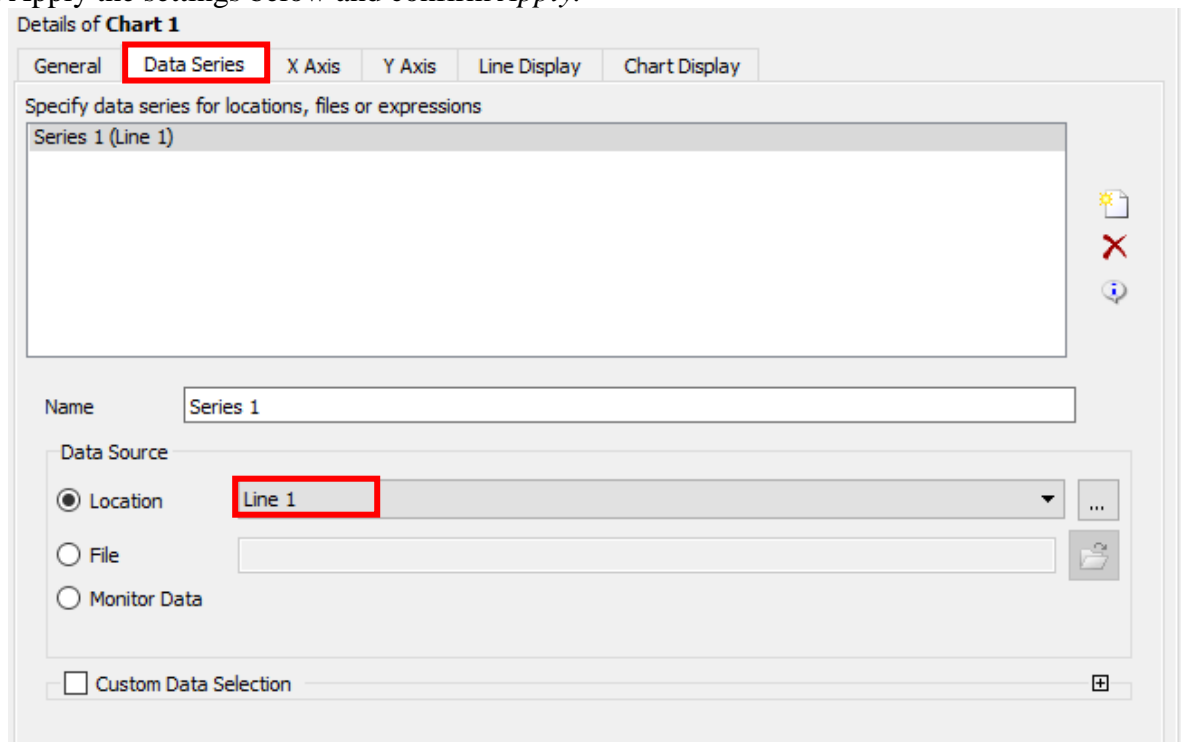
Samples 10



14) Create chart



15) Apply the settings below and confirm *Apply*.



Details of **Chart 1**

General Data Series **X Axis** Y Axis Line Display Chart Display

Data Selection

Variable **X** ...

Boundary Data ☐ Hybrid ☒ Conservative

☐ Take absolute value of data points

Axis Range

☒ Determine ranges automatically

Min -1.0 Max 1.0

☐ Logarithmic scale ☐ Invert axis

Axis Number Formatting

☒ Determine the number format automatically

Precision 3 Scientific

Axis Labels

☒ Use data for axis labels

Custom Label X Axis <units>

Details of **Chart 1**

General Data Series X Axis **Y Axis** Line Display Chart Display

Data Selection

Variable **H2O.Condensation Mass Flux** ...

Boundary Data ☒ Hybrid ☐ Conservative

☒ Take absolute value of data points

Axis Range

☒ Determine ranges automatically

Min -0.001 Max 0

☐ Logarithmic scale ☐ Invert axis

Axis Number Formatting

☒ Determine the number format automatically

Precision 3 Scientific

Axis Labels

☒ Use data for axis labels

Custom Label Y Axis <units>

16) Export the chart results to a csv file

Results to be included in the report:

- 1) Contours of distribution of mass fraction of water in the computational domain.
- 2) Contours of temperature distributions in the computational domain.
- 3) Contours of speed distribution in the computational domain.
- 4) Picture of the variable *H2O.Condensation Mass Flux* on the surface of the cold wall.
- 5) Compare analytical (1) and numerical results (results from the exported csv file) on one chart.

For the case under consideration, there is an analytical solution presented by Sparrow et al. [1]. The mass stream of condensing steam m_{cond} changes along the plate as follows

$$m_{cond} = \frac{1}{2} \sqrt{\frac{\rho \mu U_{\infty}}{x}} F(0) \quad (1)$$

where

ρ mixture density, kg/m³

μ mixture dynamic viscosity, Pa s

U_{∞} free stream velocity (at the inlet), m/s

x coordinate along the plate, m

$F(0)$ function that depends among others on the mass fraction of non-condensing gas

For analytical calculations, please take the condensate properties at saturation temperature of 100 °C, for simplicity. Required values of variables are presented in tab. 1.

Tab. 1. Values of variables that should be substituted into the equation (1)

ρ kg/m ³	μ Pa s	U_{∞} m/s	$F(0)$
958,35	0,000281745	0,1	0,05

On the chart, do not show results for $x = 0$.

3. REFERENCES

- [1] Ansys CFX Solver Modeling Guide, v 14.0, 2011.
- [2] E.M. Sparrow, W.J. Minkowycz, M. Saddy, Forced Convection condensation in the presence of noncondensables and interfacial resistance, International Journal of Heat and Mass Transfer, Vol. 10, pp. 1829-1845, 1967.