Politechnika Wrocławska

Mechanical and Power Engineering Faculty
Full-time studies

Selected problems of thermal-flow processes

Exercise no. 3

Modeling of multiphase flow

Wrocław 2020

TABLE OF CONTENTS

1. INTRODUCTION

The exercise will show how to model a simple two-phase flow. Initially, the tank is filled with air at atmospheric pressure. At one moment the valve opens and water flows into the tank at a speed of 0.1 m / s. The valve is open for 10 s. To reduce the calculation time, the case will be modeled as two-dimensional. The diagram of the analyzed case is presented in Fig. 1.

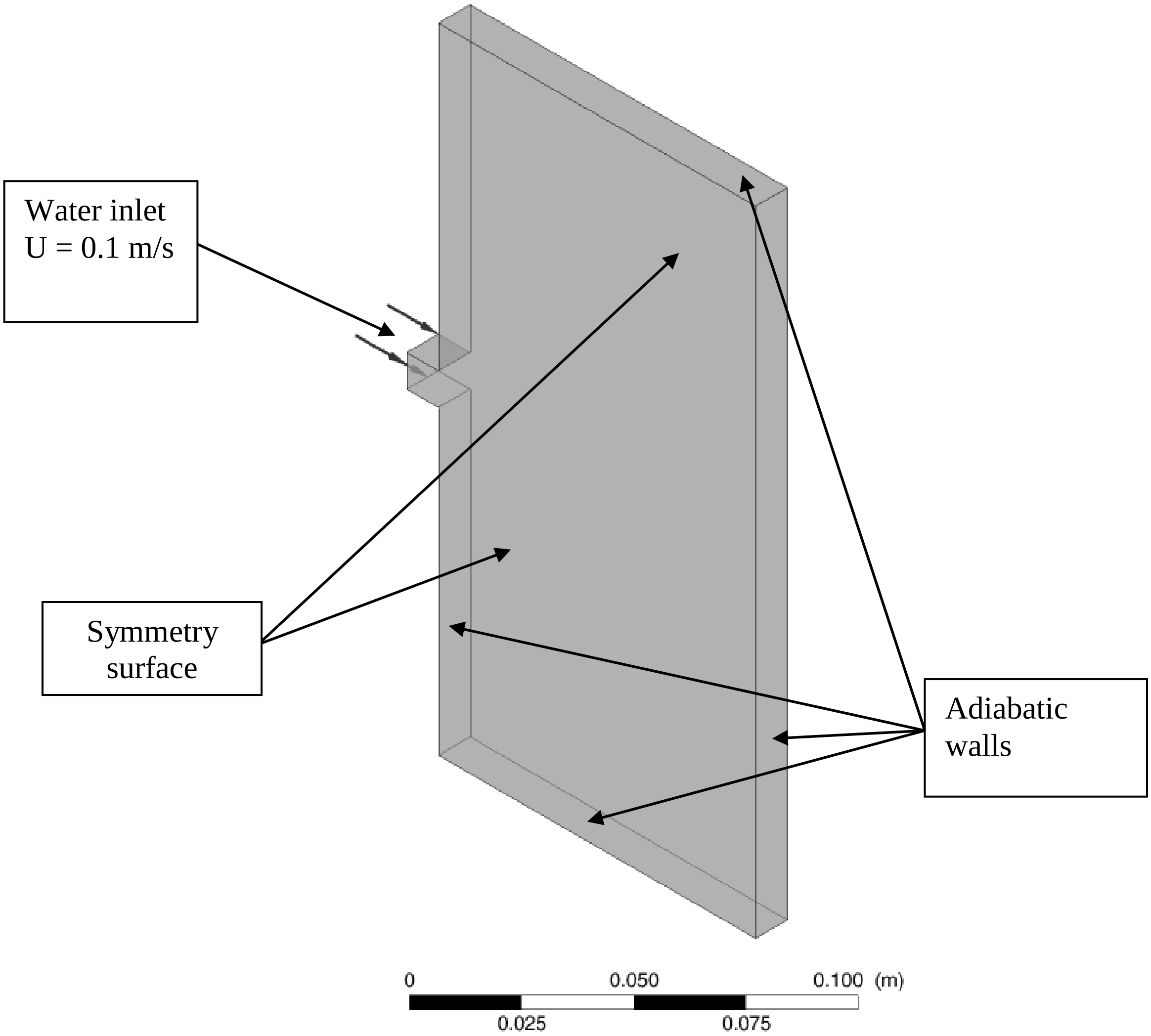


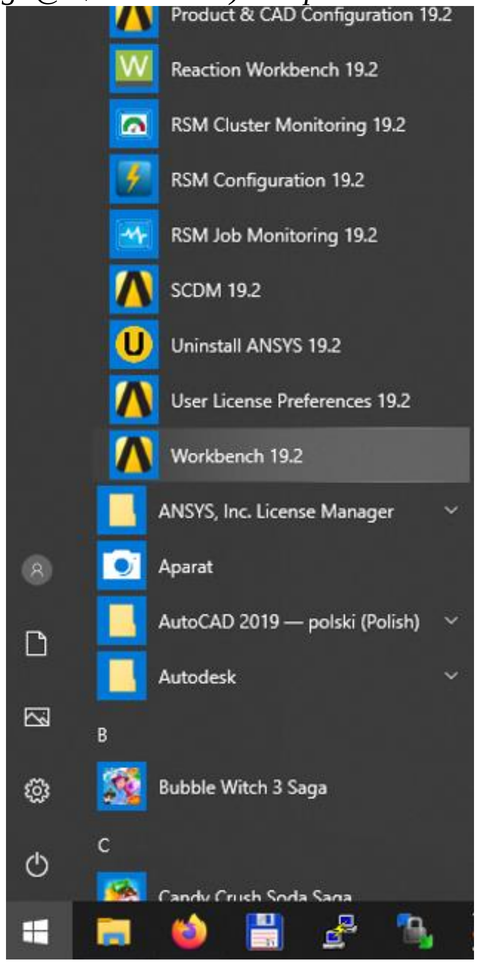
Fig. 1. Scheme of the issue of filling a tank filled with air under atmospheric pressure with water

2. TWO-DIMENSIONAL TWO-PHASE FLOW OF WATER AND AIR

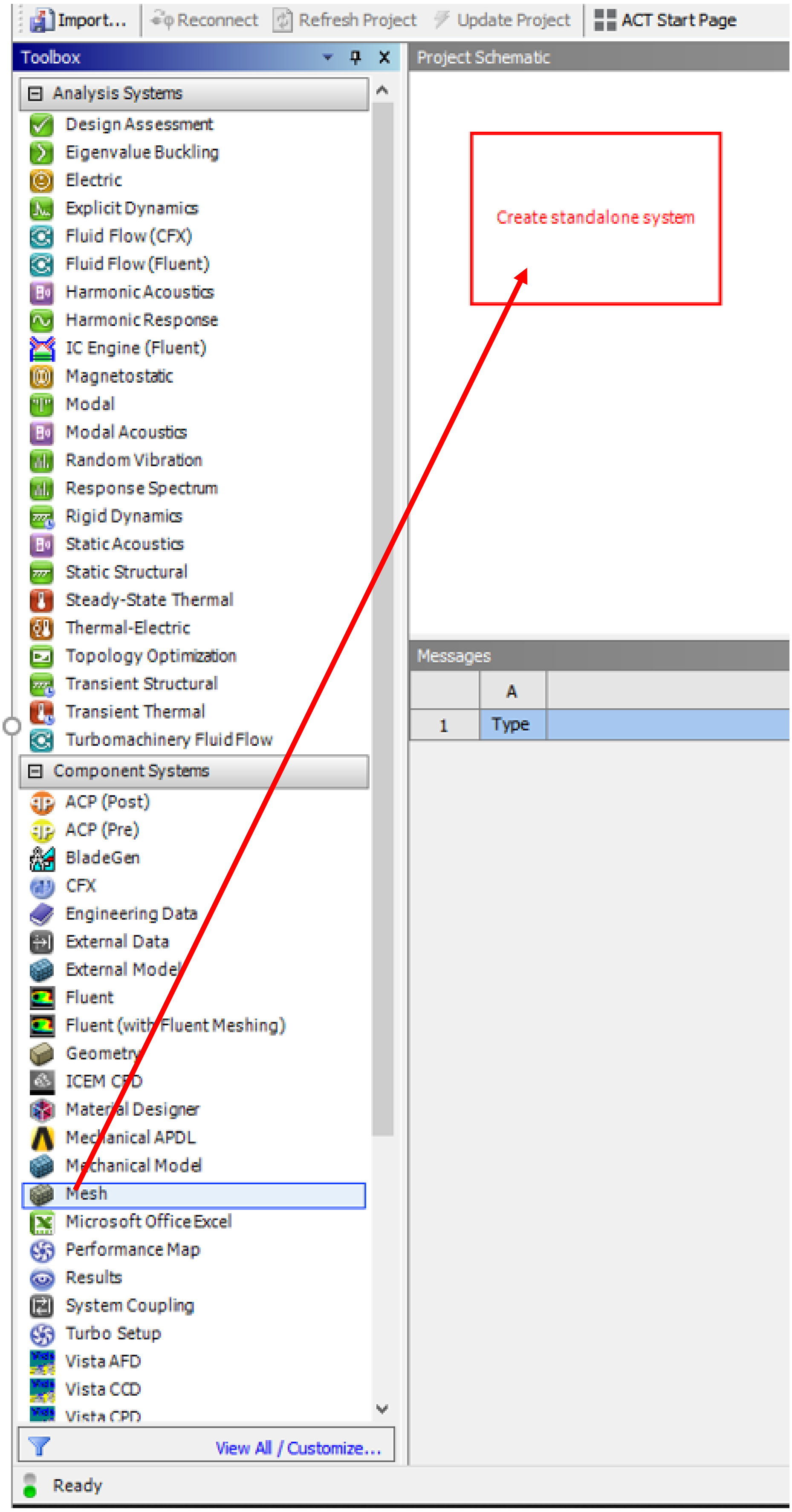
2.1. GEOMETRY

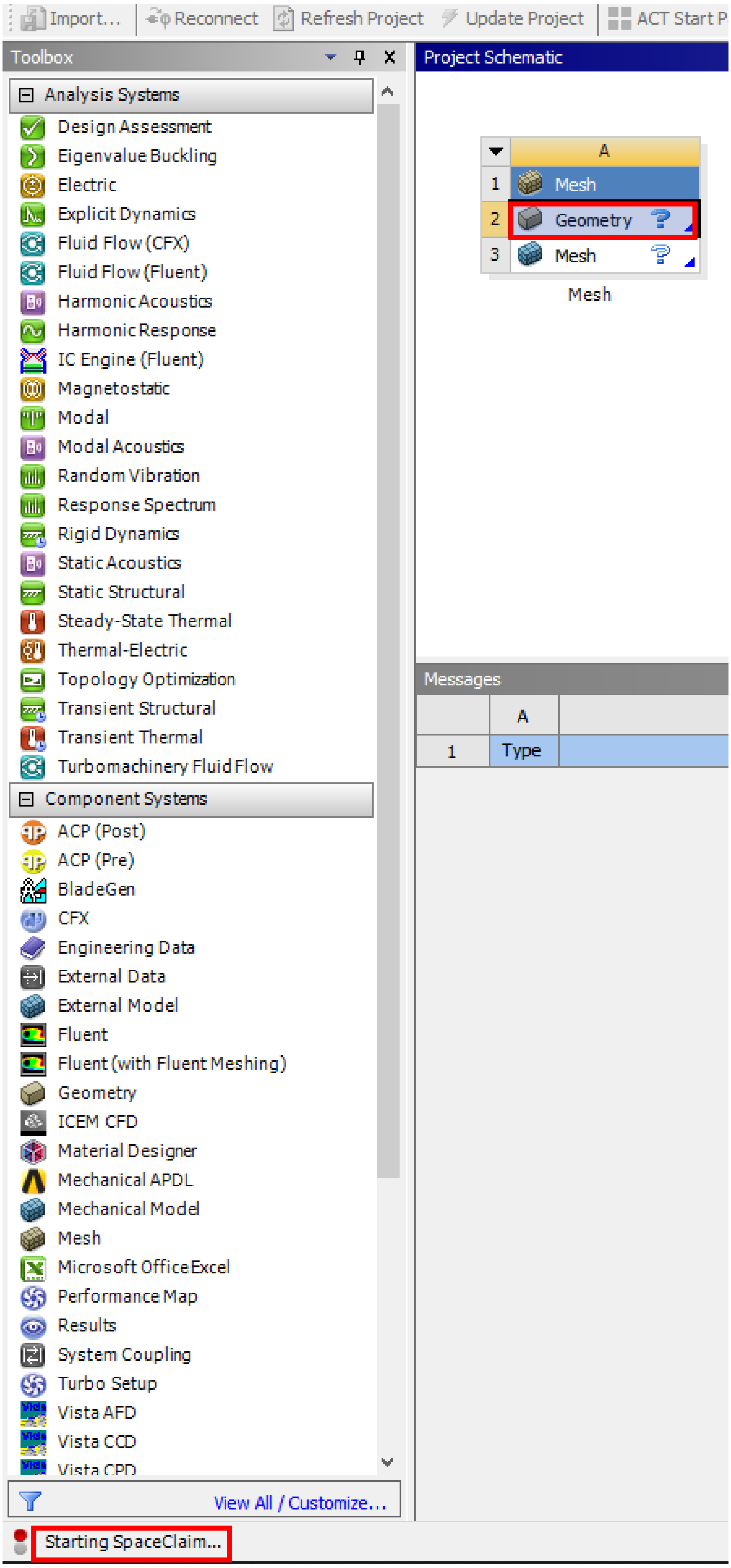
Do the following:

1) Open Ansys Workbench and save project as Ex3 in folder Ex3 (*File->Save As*).
RULE OF THUMB NO. 1: *We 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. @#$%^&* etc.) and polish marks*



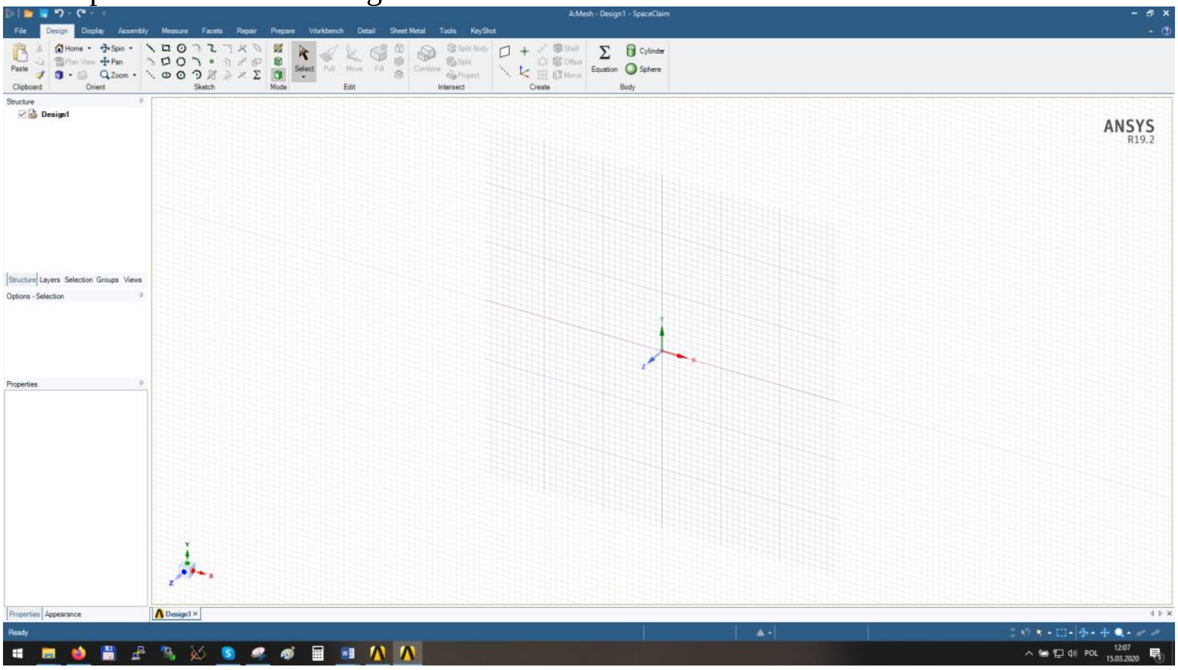
2) Select the *Mesh* module and open *Spaceclaim*. To do this, hold the left mouse button (LMB) on the *Mesh* module and drag it to the *Project Schematic* field. Then double-click LMB on *Geometry* to start the *Spaceclaim* program in which the geometry will be created. Note that in the lower left corner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

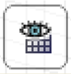

Import...
Reconnect
Refresh Project
Update Project
ACT Start Page
Toolbox
Project Schematic
Analysis Systems
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (CFX)
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
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
View All / Customize...
Ready
Create standalone system
Messages
A
1
Type


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
View All / Customize…
Starting SpaceClaim…
Project Schematic
A
1 Mesh
2 Geometry
3 Mesh
Mesh
Messages
A
1 Type

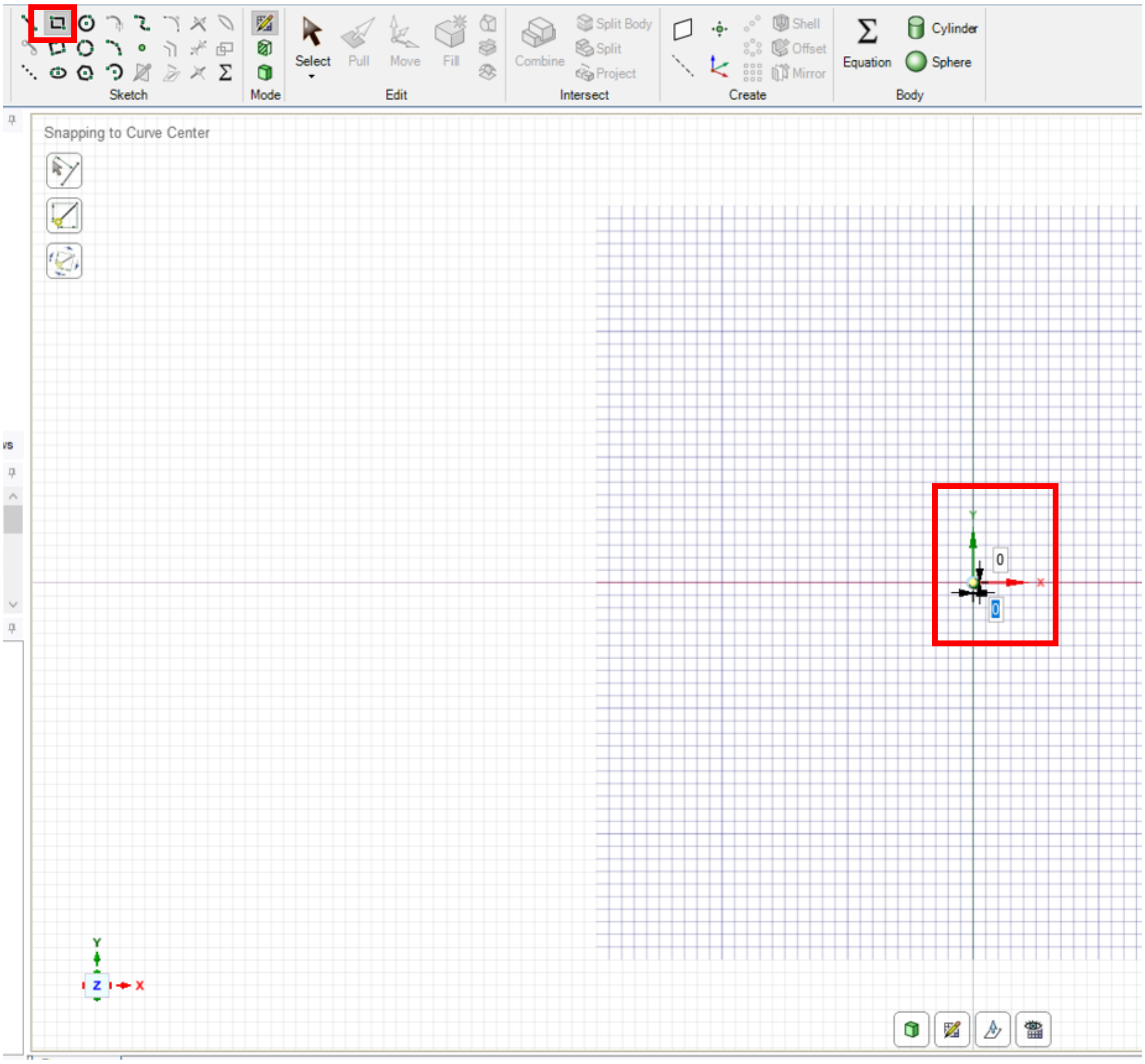
3) Click LMB *Select New Sketch*  to select the drawing plane.

Select plane X-Y as in the figure below.

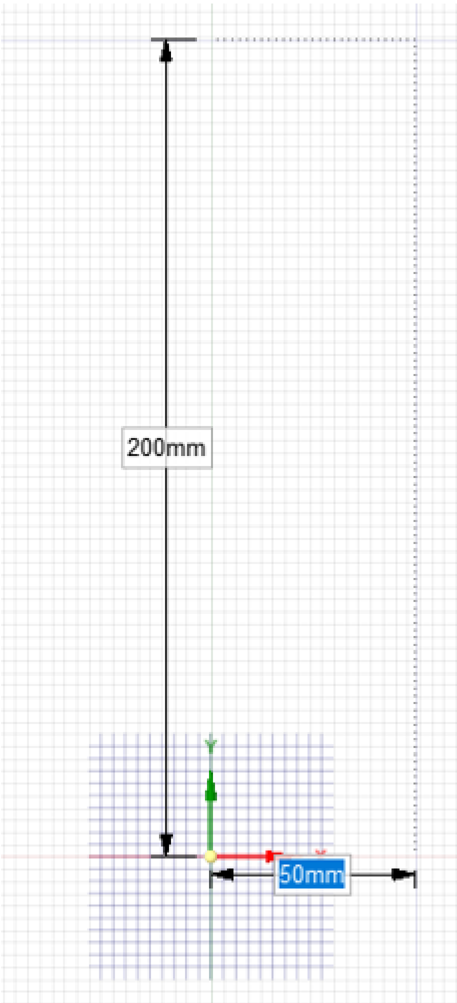


4) Click *Plan View*  to rotate the drawing plane parallel to the screen (you can also do this by pressing *Shift* + *v*).

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 *Shift* key – two dimensions editing fields will appear as shown below.



You can switch between the dimension edit fields with the *Tab* key. Set the horizontal dimension to 50 and the vertical dimension to 200 mm and press *Enter*.

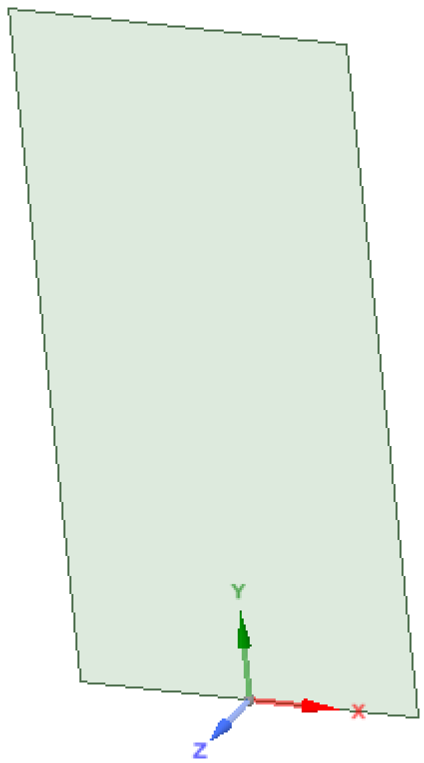


The program will proceed to draw a rectangle. Enter 100 mm for the horizontal dimension and 200 for the vertical dimension and confirm with *Enter*.

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*.

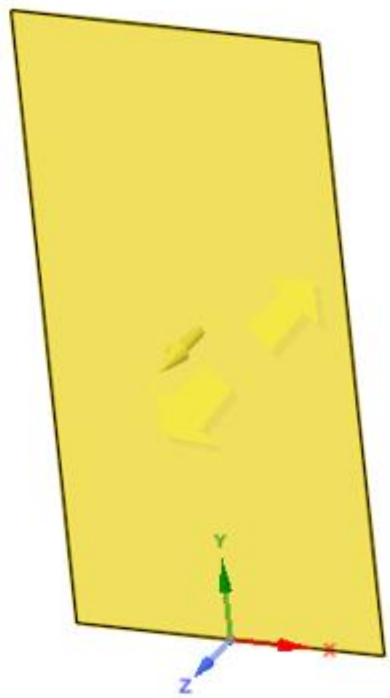
6) To exit the rectangle drawing command, press *Esc* and LMB, click the *Return to 3D mode* icon

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

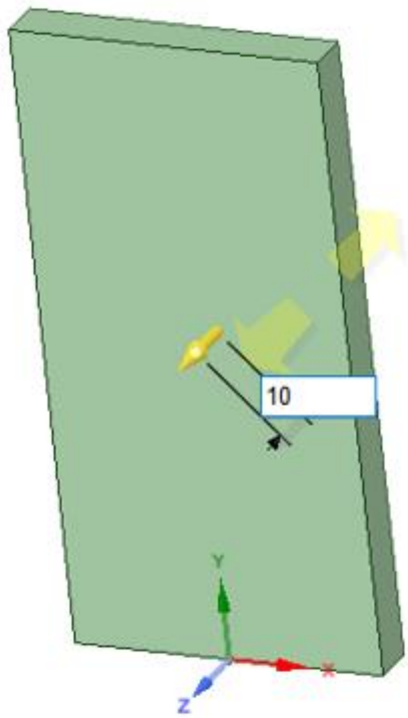


8) Choose *Pull*

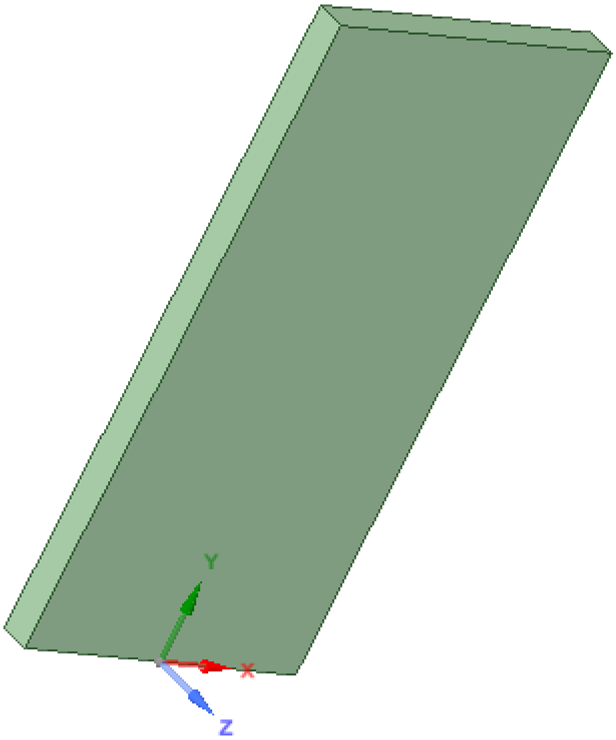
Then position the cursor as shown below



By moving the cursor while pressing LMB you will notice the changing dimension of the pipe length. Type 10 mm and confirm with *Enter* (you may have to do it while holding LMB).

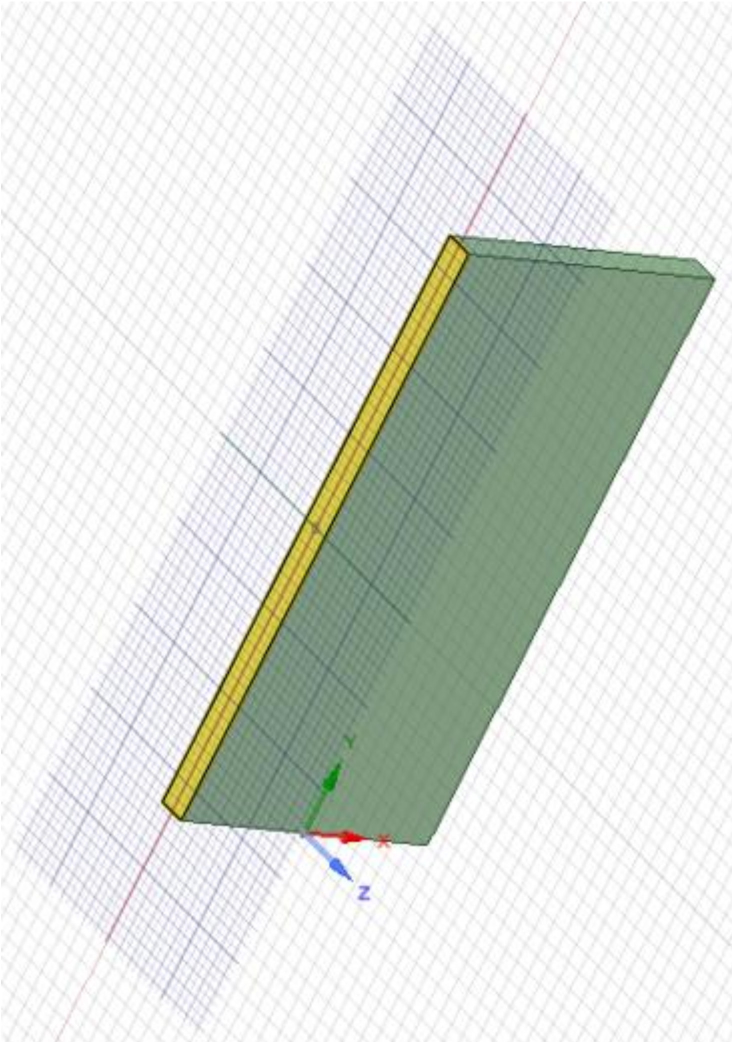


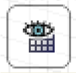
9) Use the scroll button to rotate the model to see the YZ plane



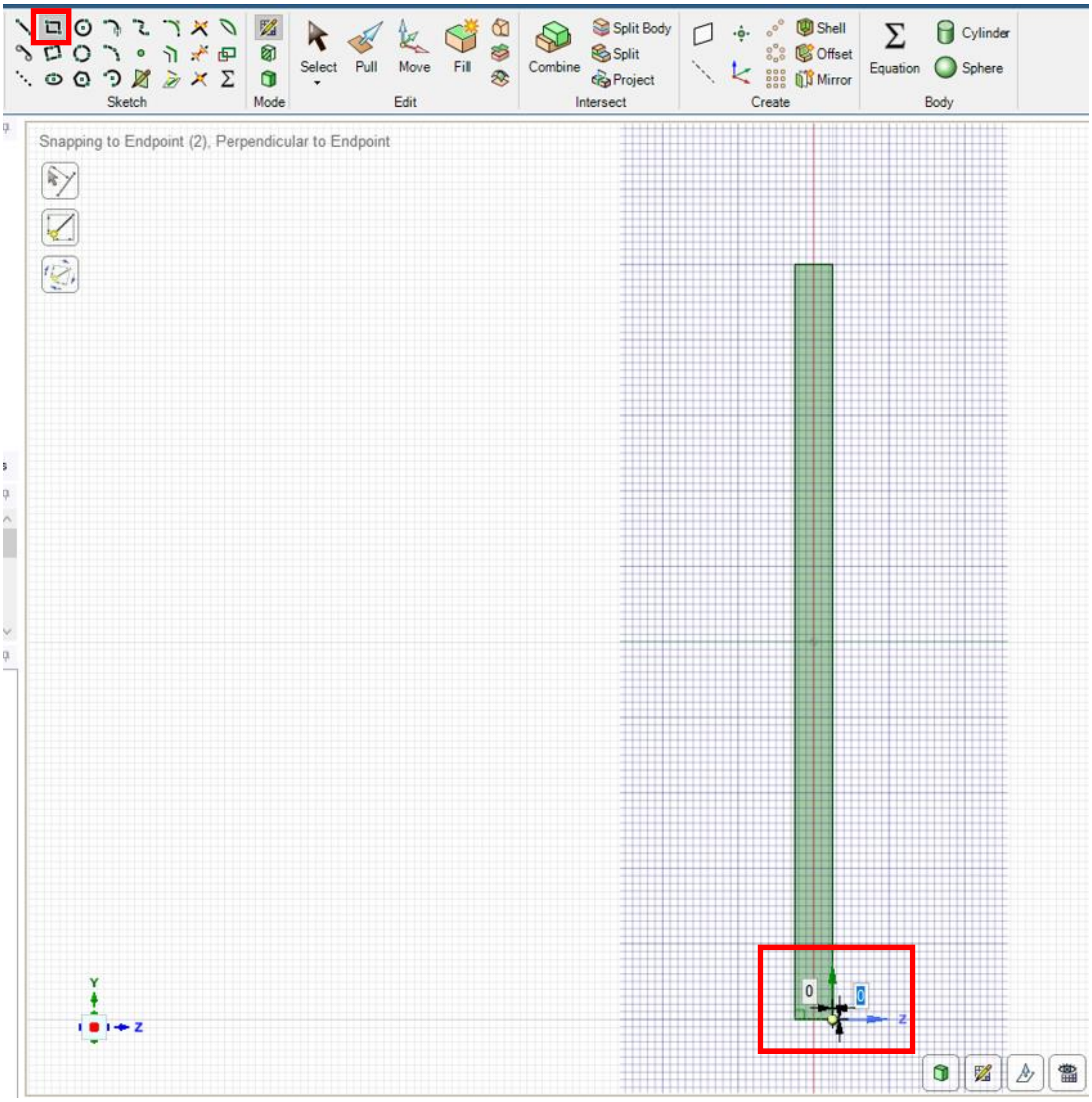
10) Click LMB *Select New Sketch*  to select new sketch plane.

Select the Y-Z plane as shown below.

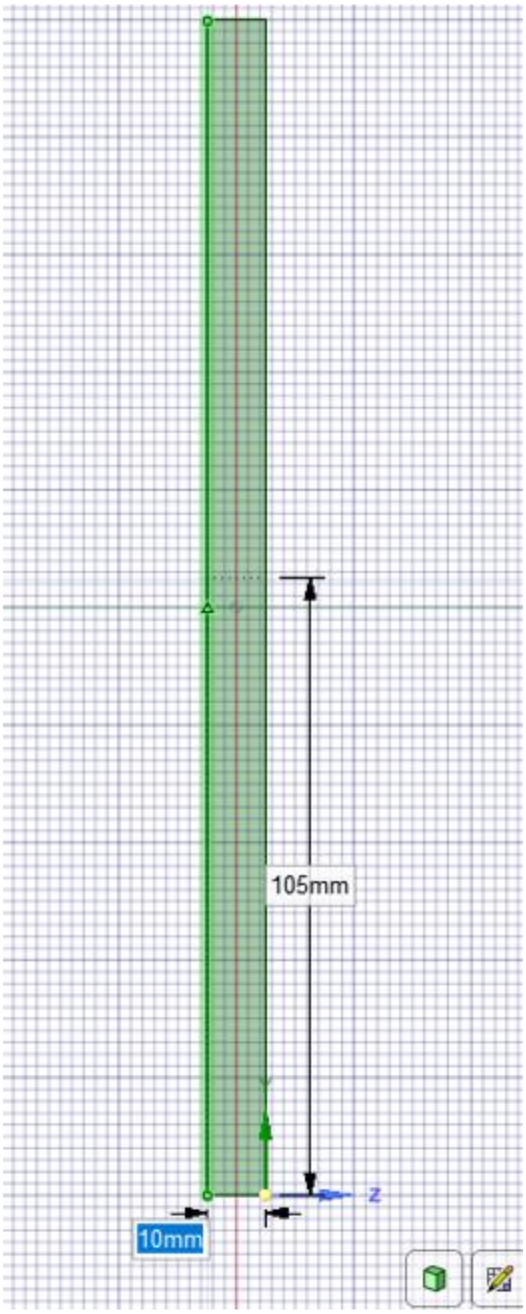


11) Click *Plan View*  to rotate the drawing plane parallel to the screen (you can also do this by pressing *Shift* + *v*).

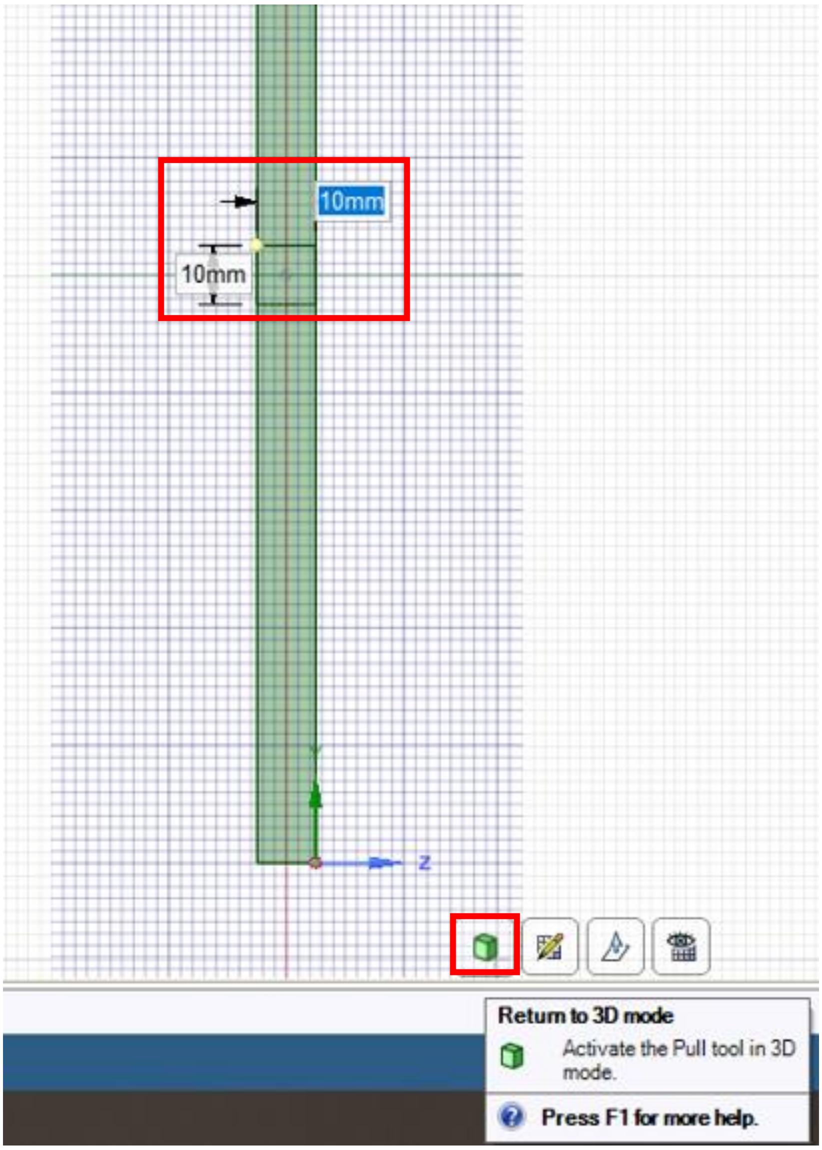
12) Select the rectangle drawing icon and move the cursor to the beginning of the coordinate system, then press the *Shift* key



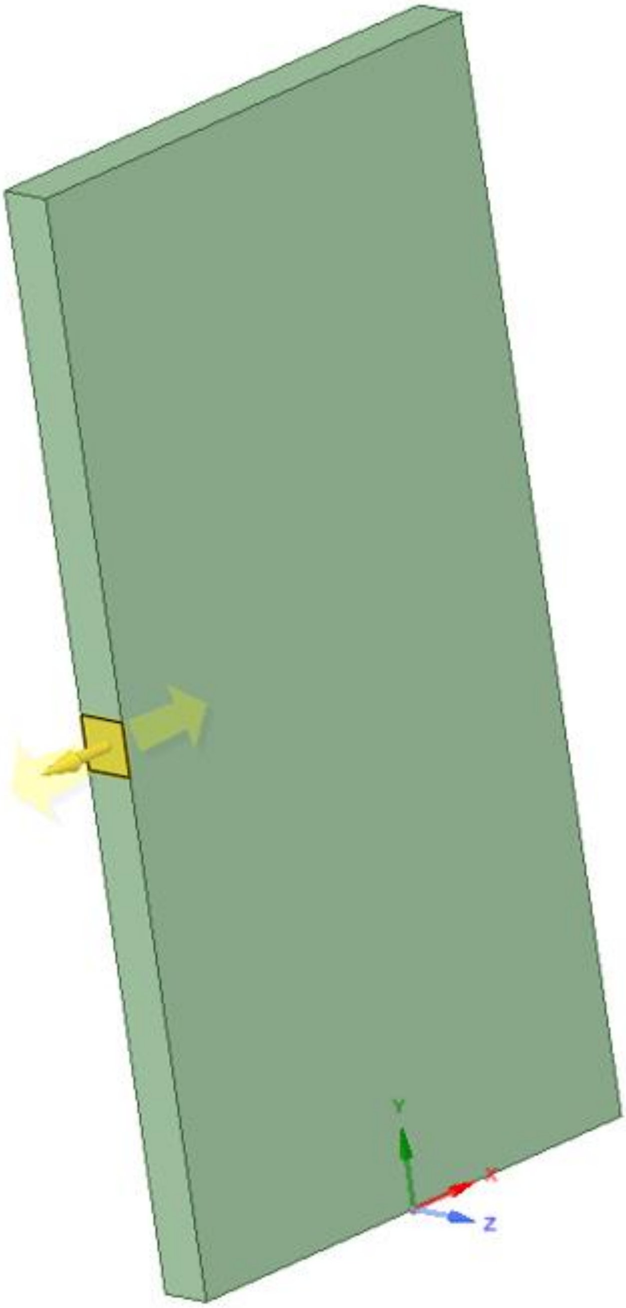
13) Apply the dimensions as below and confirm *Enter* (switch between dimensions with the *Tab* key)



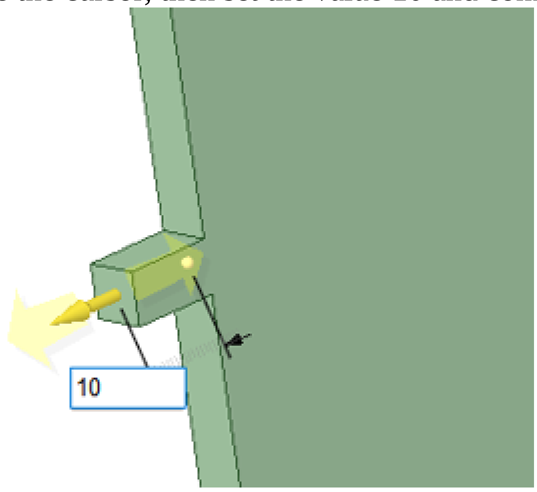
Then apply the following dimensions and proceed to 3D drawing



14) Position the cursor as shown below



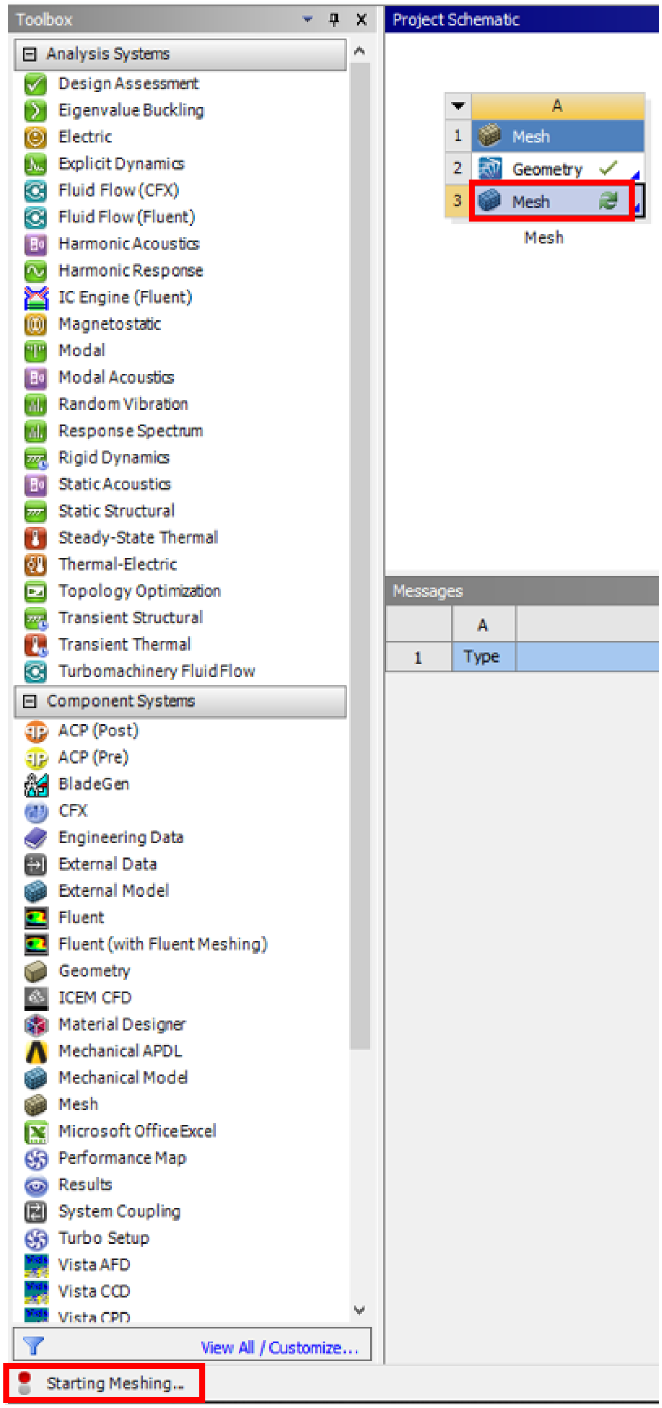
15) Press LMB and move the cursor, then set the value 10 and confirm *Enter*



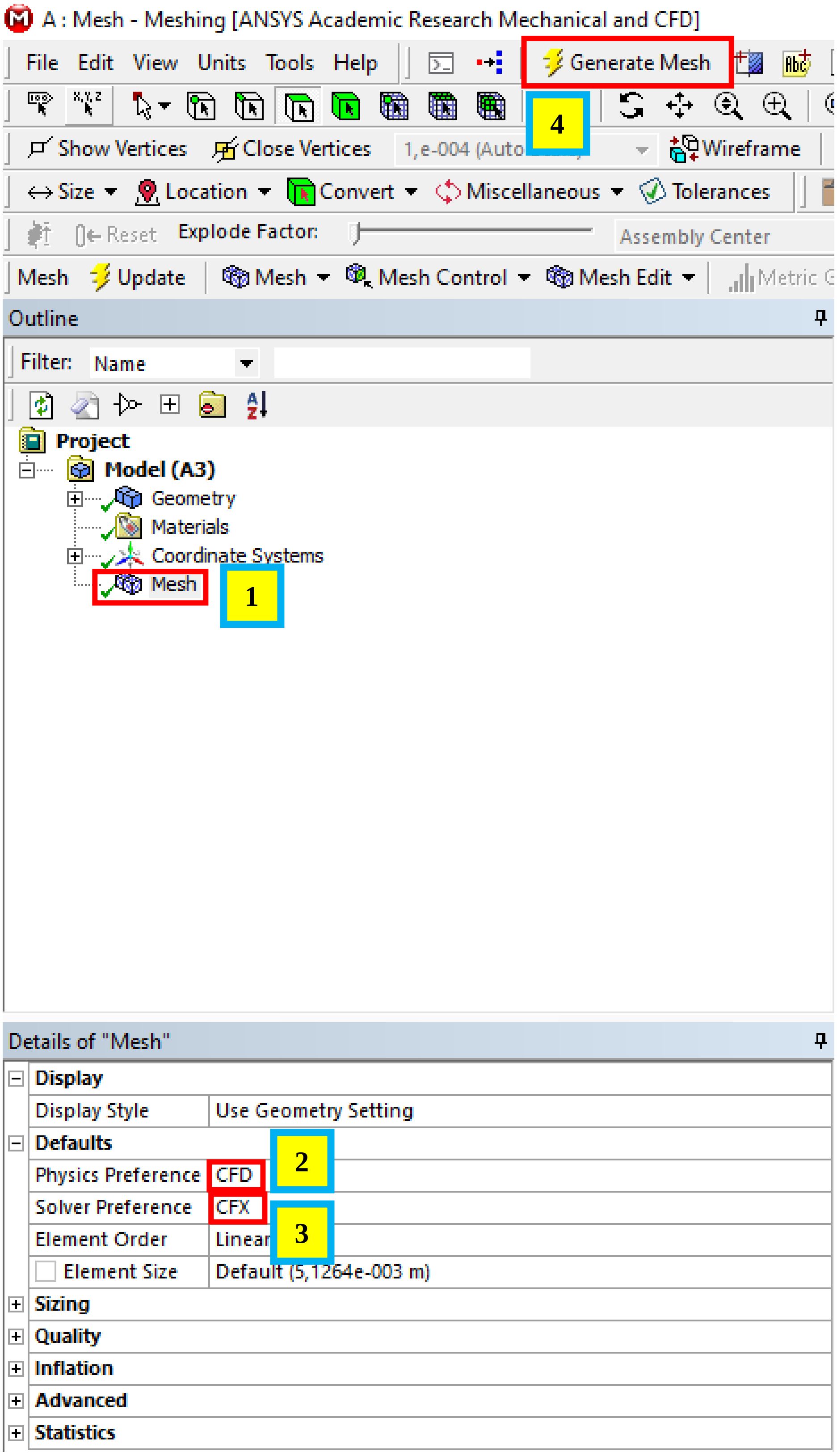
16) Close *Spaceclaim* and save project in *Workbench* using *Ctrl + s*

2.2. NUMERICAL MESH

1) To do the mesh open *Ansys Meshing* by double-click of LMB on *Mesh*

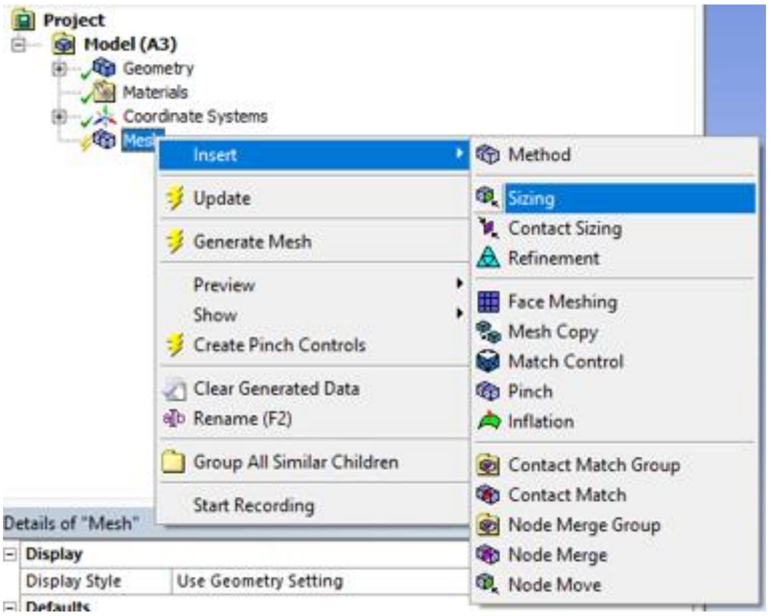


2) In *Ansys Meshing*: 1) click *Mesh*, 2) change *Physisc Preference* into *CFD*, 3) change *Solver Preference* into *CFX*, 4) Click LMB on *Generate Mesh*

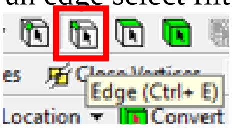


The default grid is not valid. The grid should be edited.

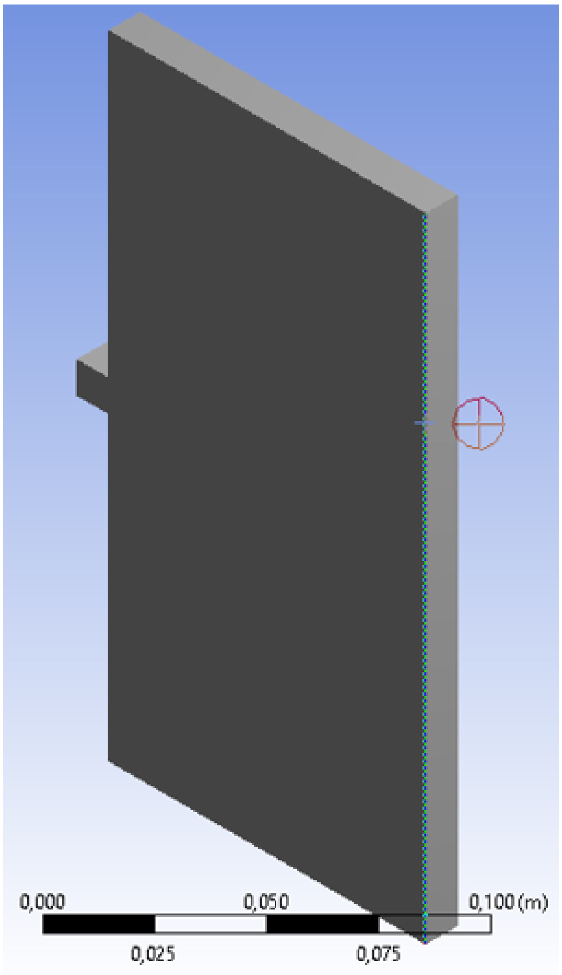
3) In *Ansys Meshing* press right button of the mouse (RMB) on *Mesh* and select *Insert->Sizing*



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



With the *Ctrl* key pressed, select two edges as shown in the figure below and approve *Geometry-> Apply* (if you don't see *Apply*, click LMB in the yellow field).



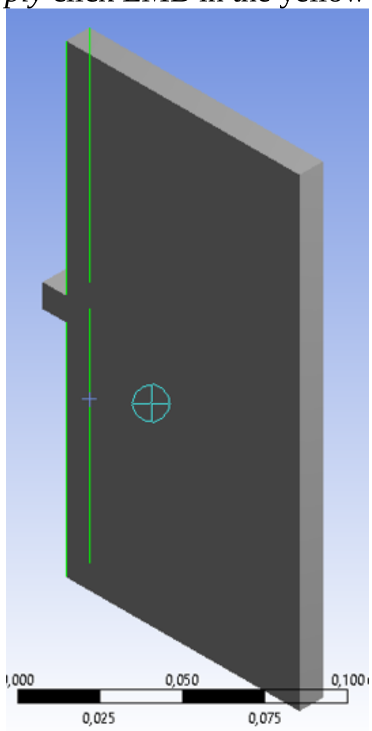
Details of "Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	Apply / Cancel
Definition	
Suppressed	No
Type	Element Size
Element Size	Default (5,1264e-003 m)
Advanced	
Defeature Size	Default (2,5632e-005 m)
Behavior	Soft
Growth Rate	Default (1,2)
Capture Curvature	No
Capture Proximity	No

Change *Definition* as below

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	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

Click *Generate Mesh* and check the generated grid (if the grid is not visible, 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 four edges as shown below and confirm *Geometry->Apply* (if you don't see *Apply* click LMB in the yellow field).

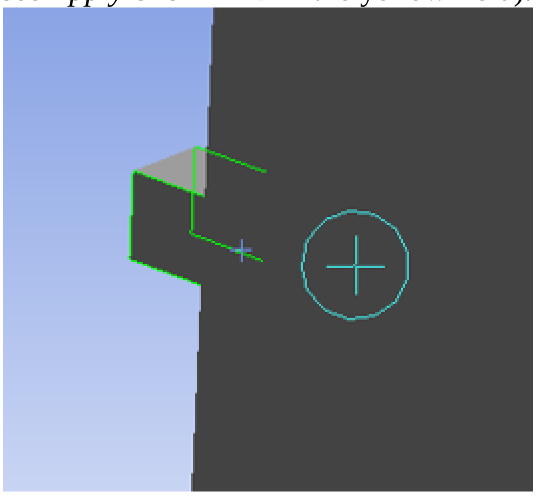


Apply the following settings

Details of "Edge Sizing 2" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	45
Advanced	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

Click *Generate Mesh* and check the generated grid (if the grid is not visible, click LMB on *Mesh* in the tree on the left).

5) In *Ansys Meshing* press RMB on *Mesh* and select *Insert->Sizing*. While holding down the *Ctrl* key, select six edges as shown below and confirm *Geometry-> Apply* (if you don't see *Apply* click LMB in the yellow field).

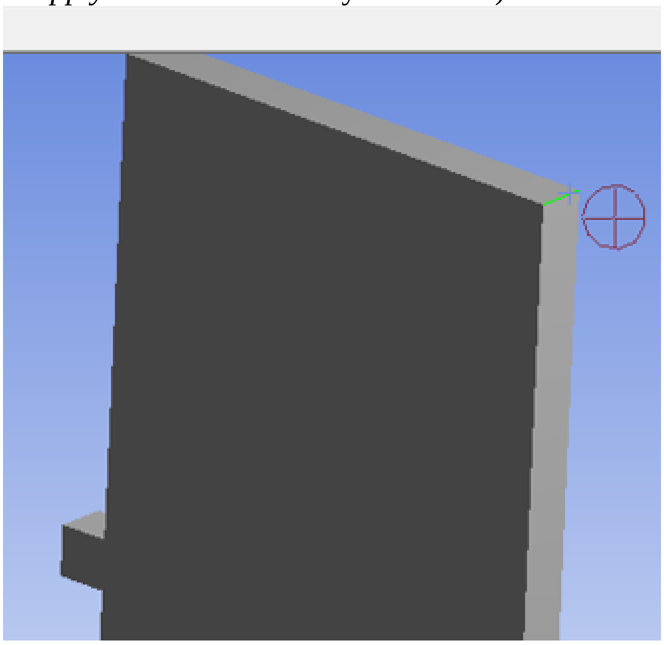


Apply the following settings

Details of "Edge Sizing 3" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	6 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	10
Advanced	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

Click *Generate Mesh* and check the generated grid (if the grid is not visible, click LMB on *Mesh* in the tree on the left).

6) In *Ansys Meshing* press RMB on *Mesh* and select *Insert->Sizing*. While holding down the *Ctrl* key, select edge as shown below and confirm *Geometry-> Apply* (if you don't see *Apply* click LMB in the yellow field).

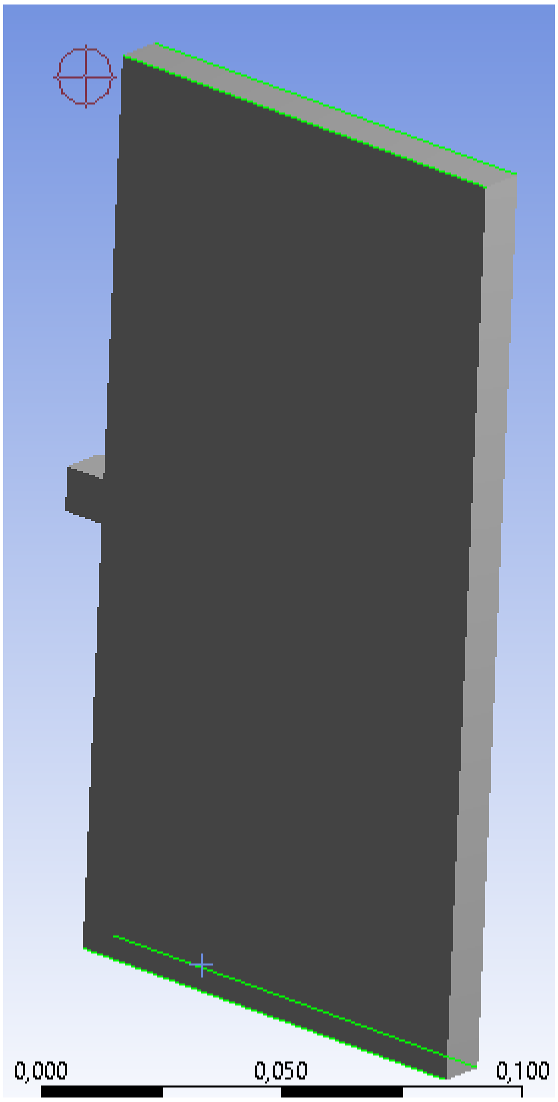


Apply the following settings

Details of "Edge Sizing 4" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	1
Advanced	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

Click Generate Mesh and check the generated grid (if the grid is not visible, click LMB on Mesh in the tree on the left).

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

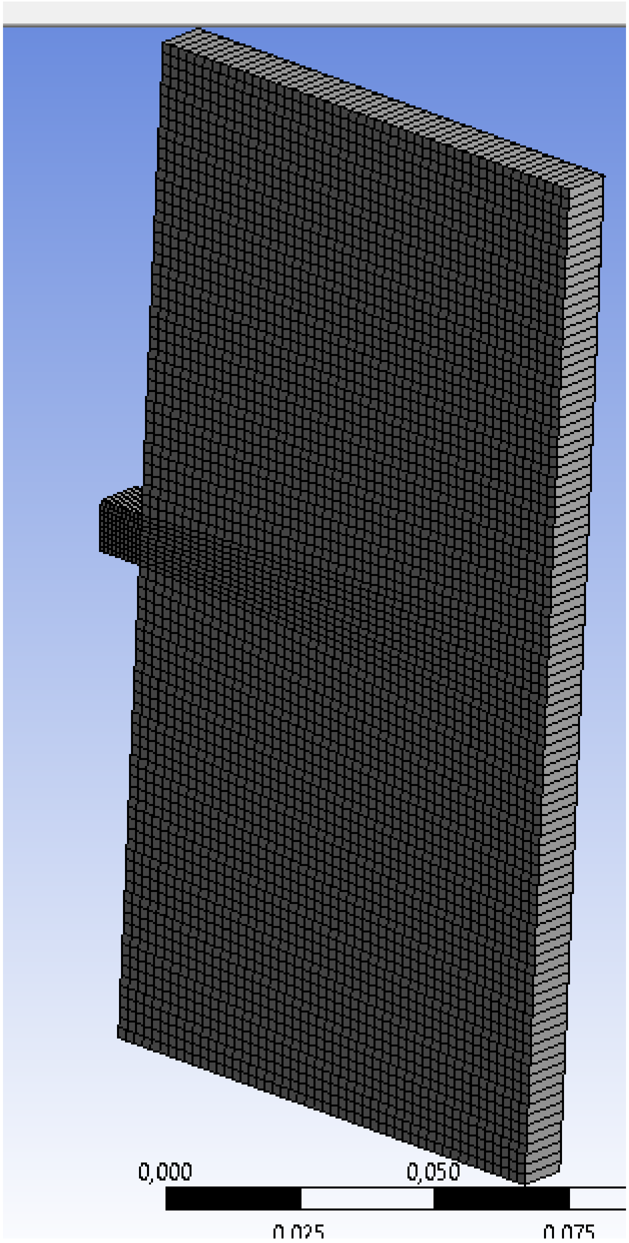


Apply the following settings

Details of "Edge Sizing 5" - Sizing

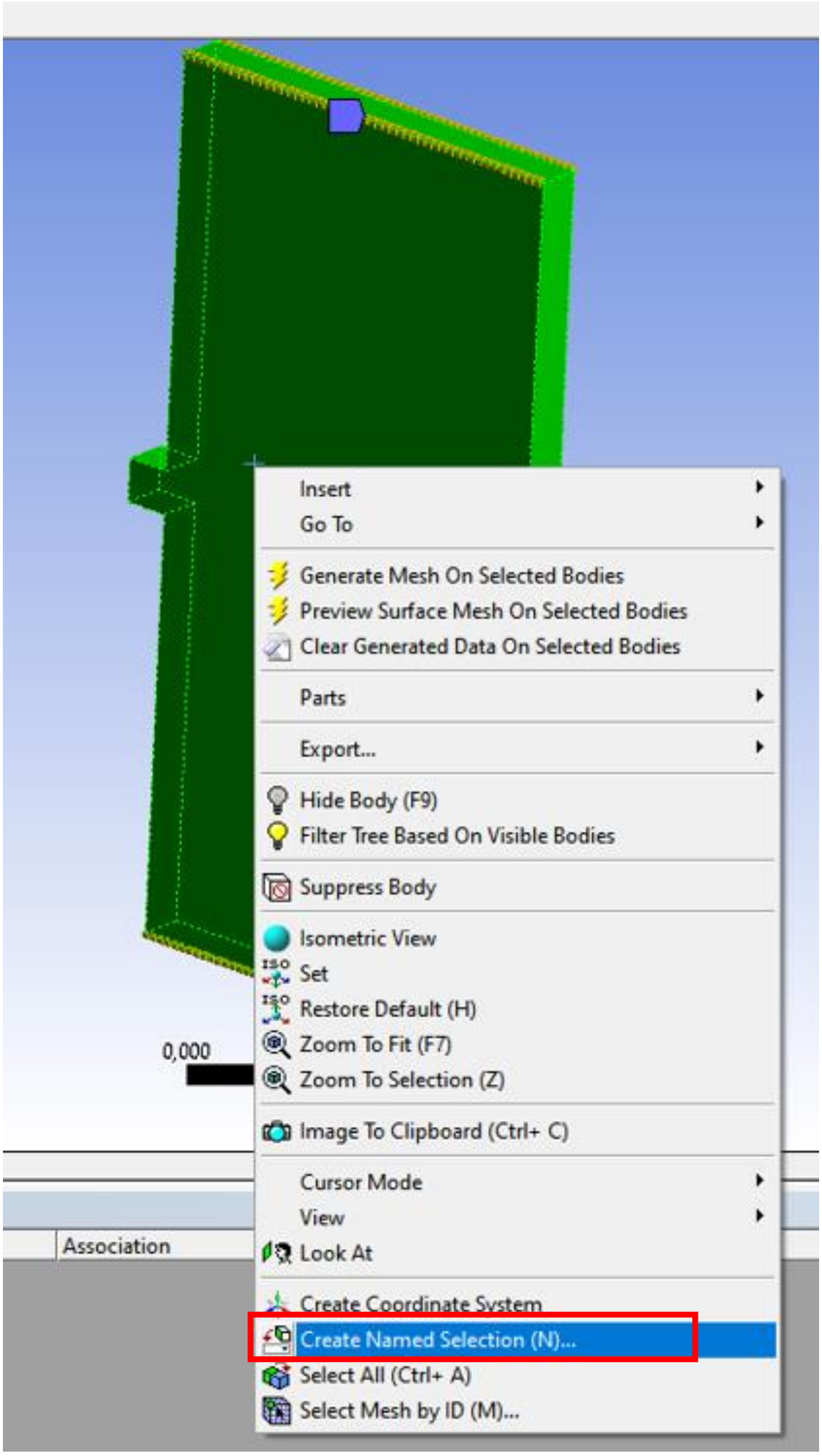
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	50
Advanced	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

8) Click Generate Mesh and check the generated grid (if the grid is not visible, click LMB on Mesh in the tree on the left). Numerical mesh is correct.

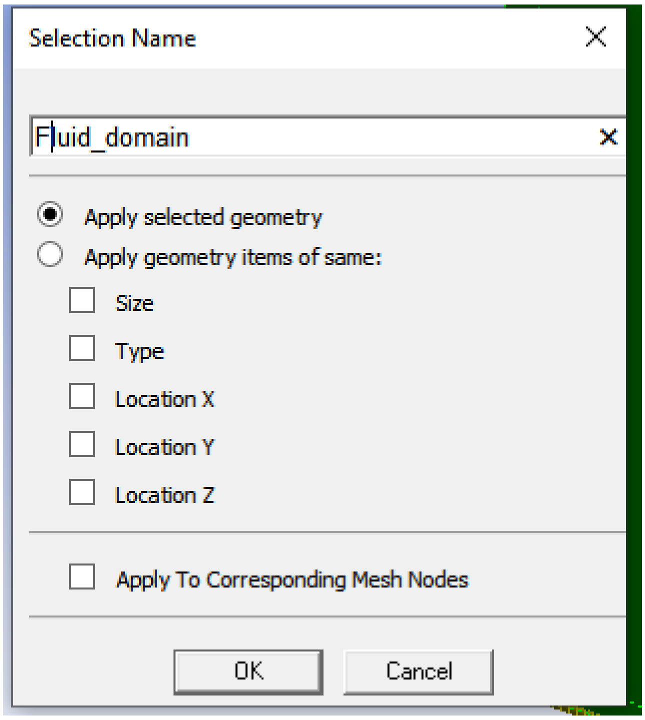


9) The last step is to name the volumes and surfaces. Select the LMB solid selection filter

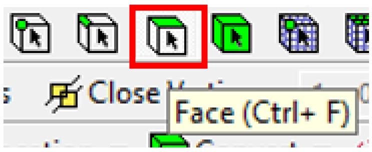
Select the LMB pipe, then click RMB and select *Create Named Selection*



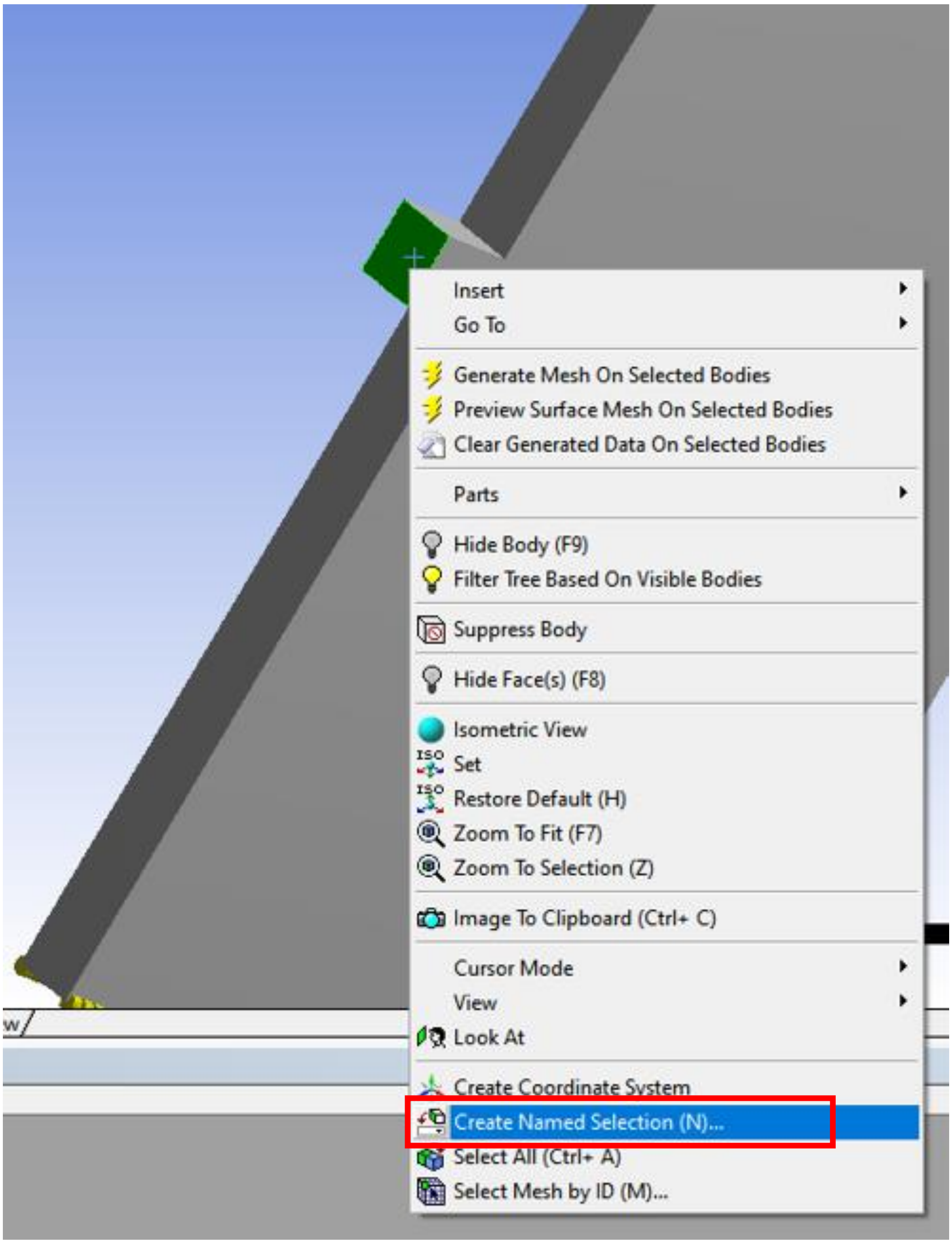
As a name type *Fluid_domain*



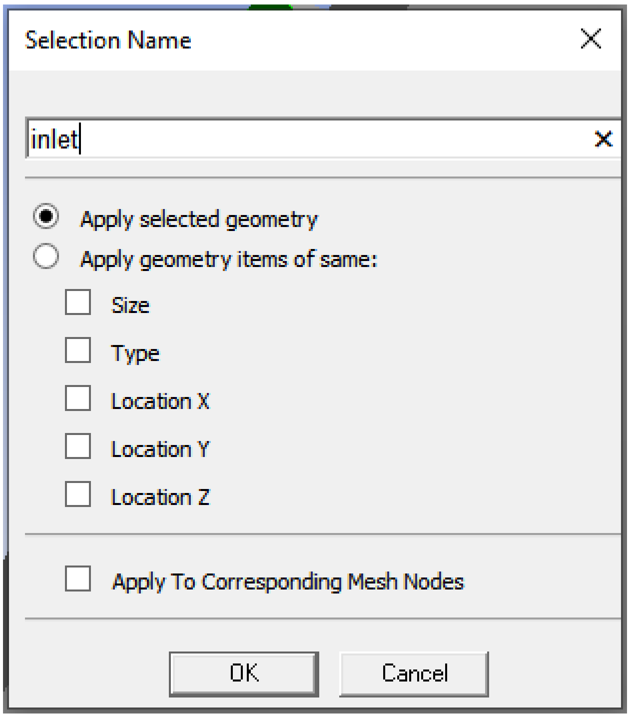
10) Then change the filter to face selection



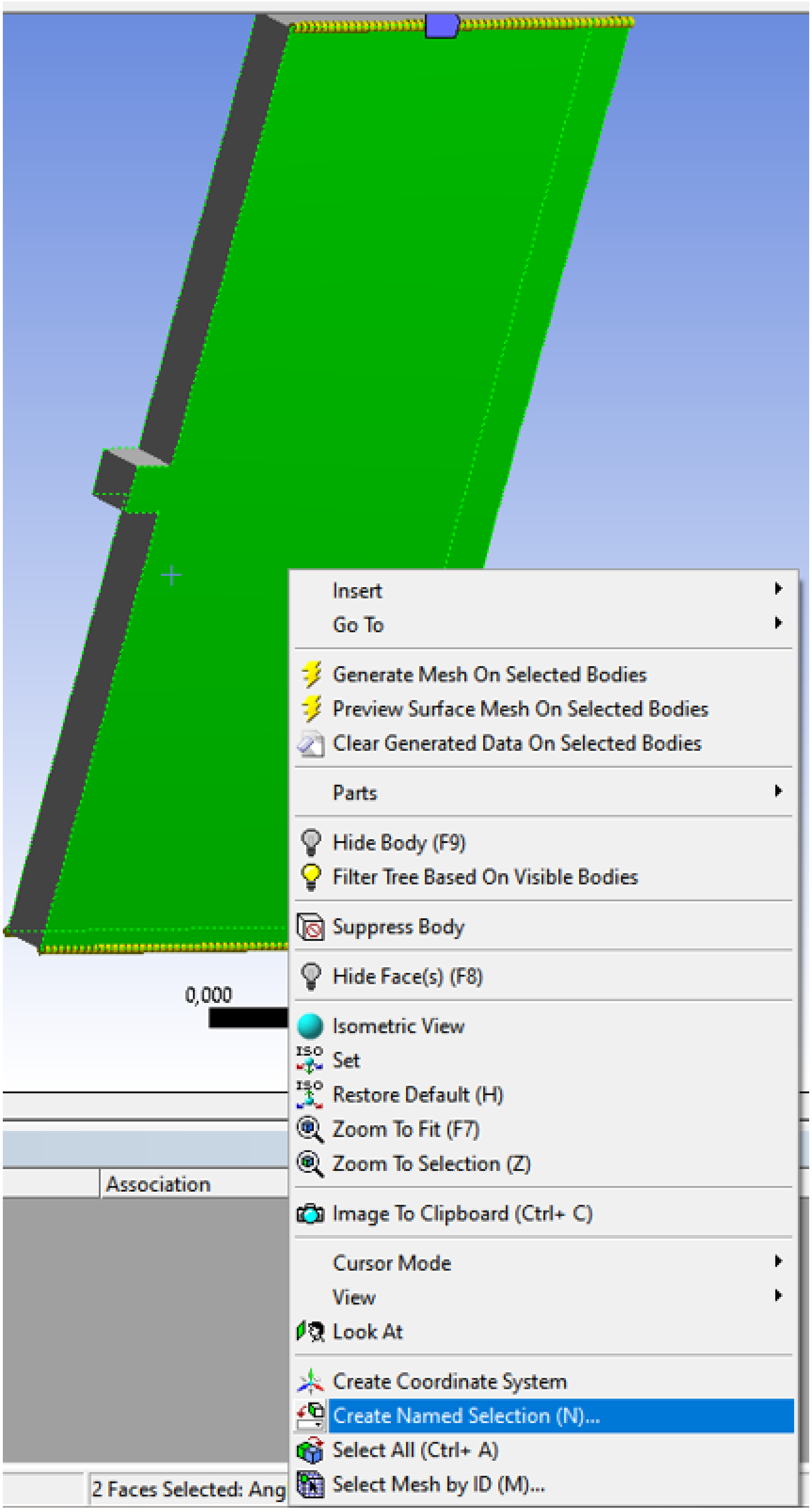
LMB point to the outer surface of the tab and click RMB, then select *Create Named Selection*



Set name as *inlet*



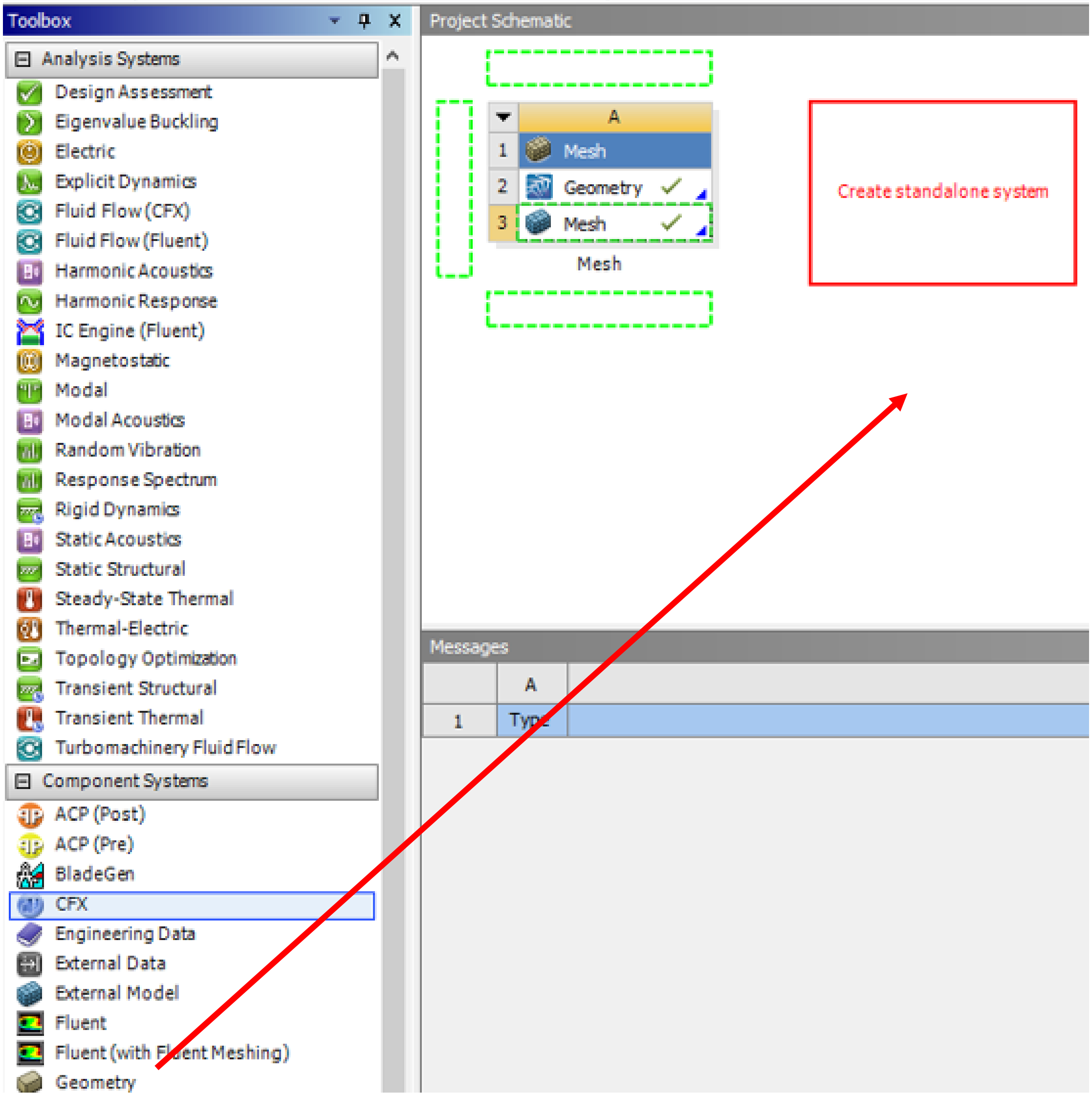
11) Similarly create *sym* for the two flat surfaces



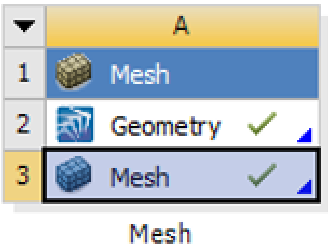
12) Close *Ansys Meshing* and save project in *Workbench.*

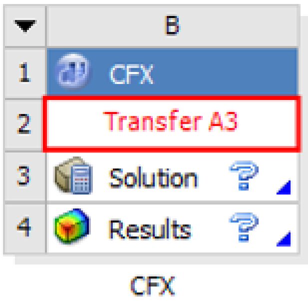
2.3. Numerical model

1) Insert *CFX* module

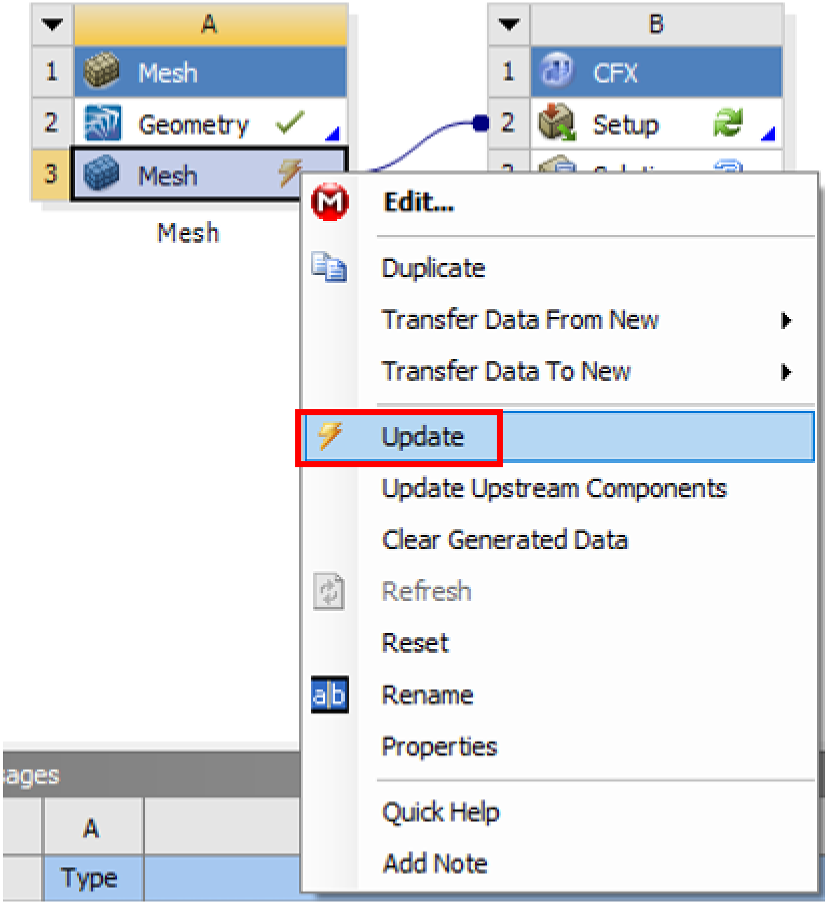


To connect the *Mesh* module with *CFX*, grab the LMB *Mesh* (below) and drag it to *Setup* until the *Transfer A3* box appears, and then release the LMB - the connection has been created.

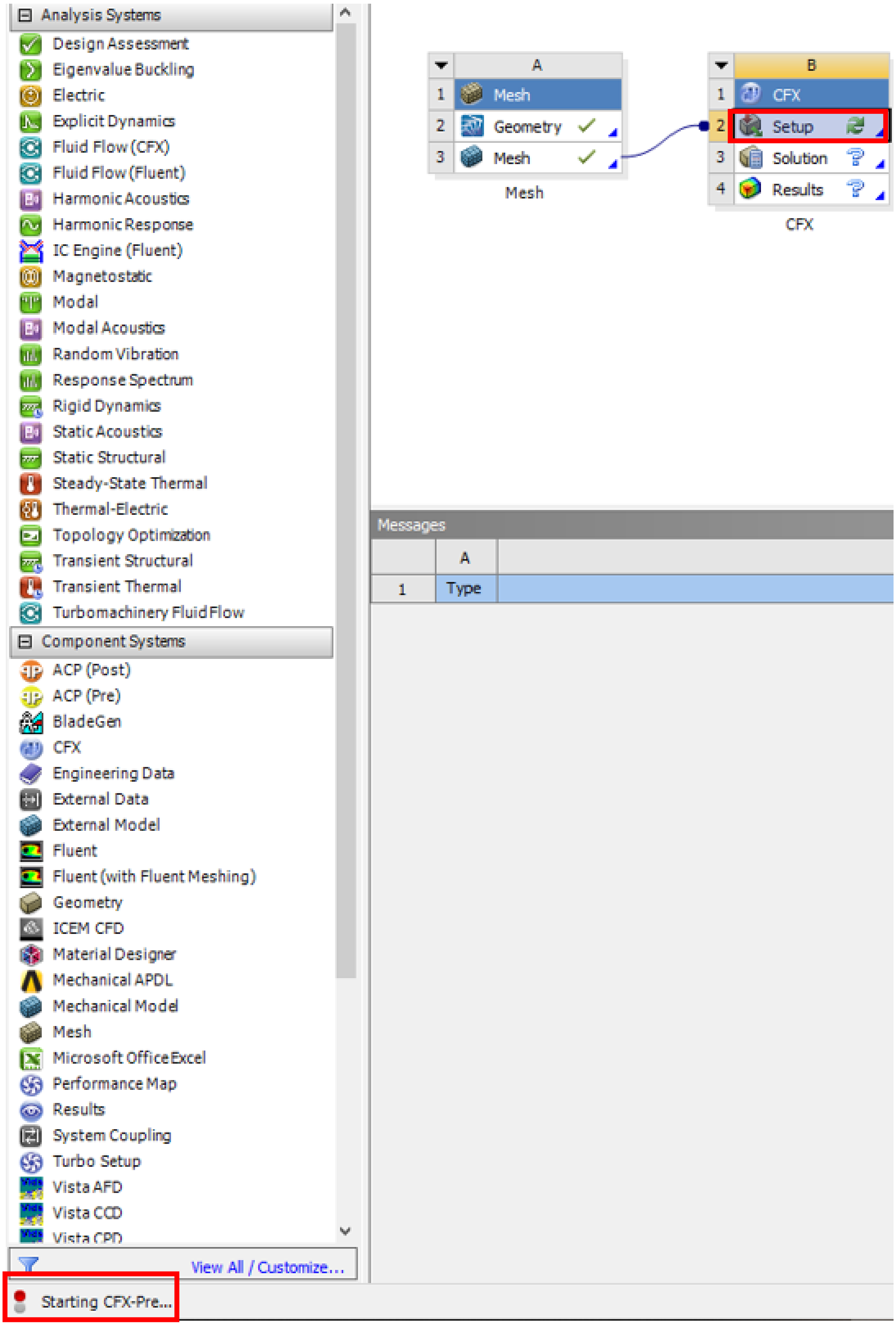




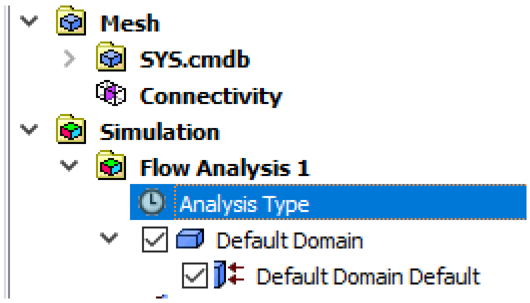
Click RMB on *Mesh* and select *Update*



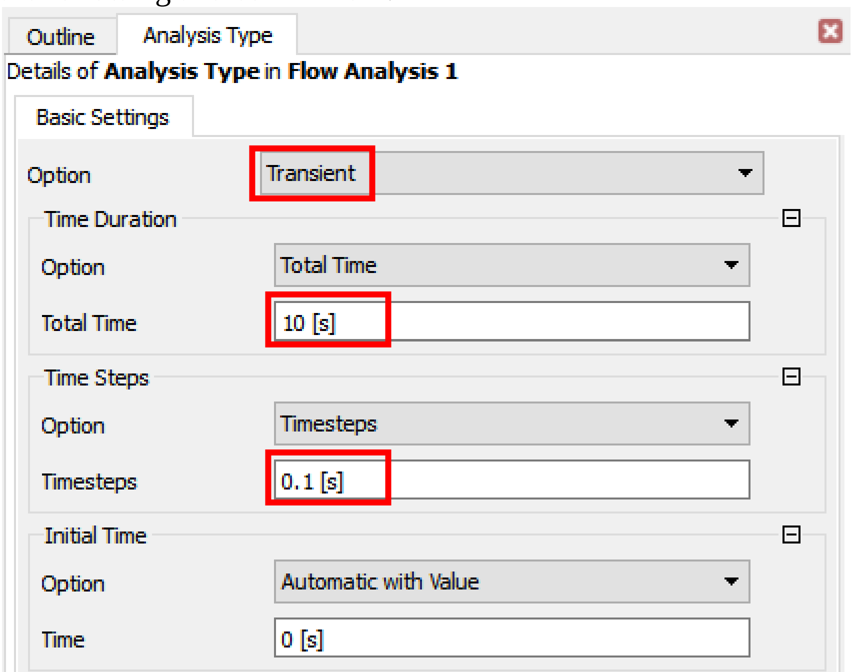
Double-click *Setup* to run *Ansys CFX*



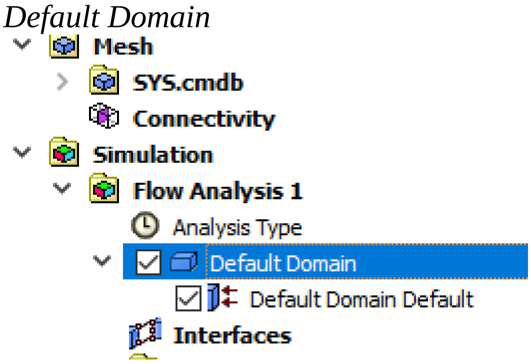
2) Open *Analysis Type* by double-clicking LMB



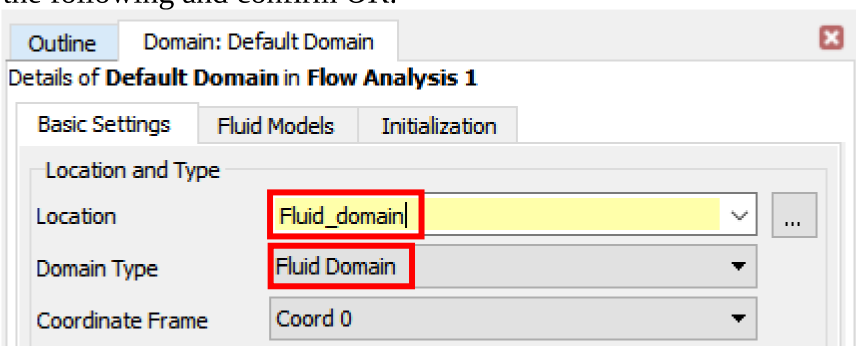
Apply the following and confirm *OK*.



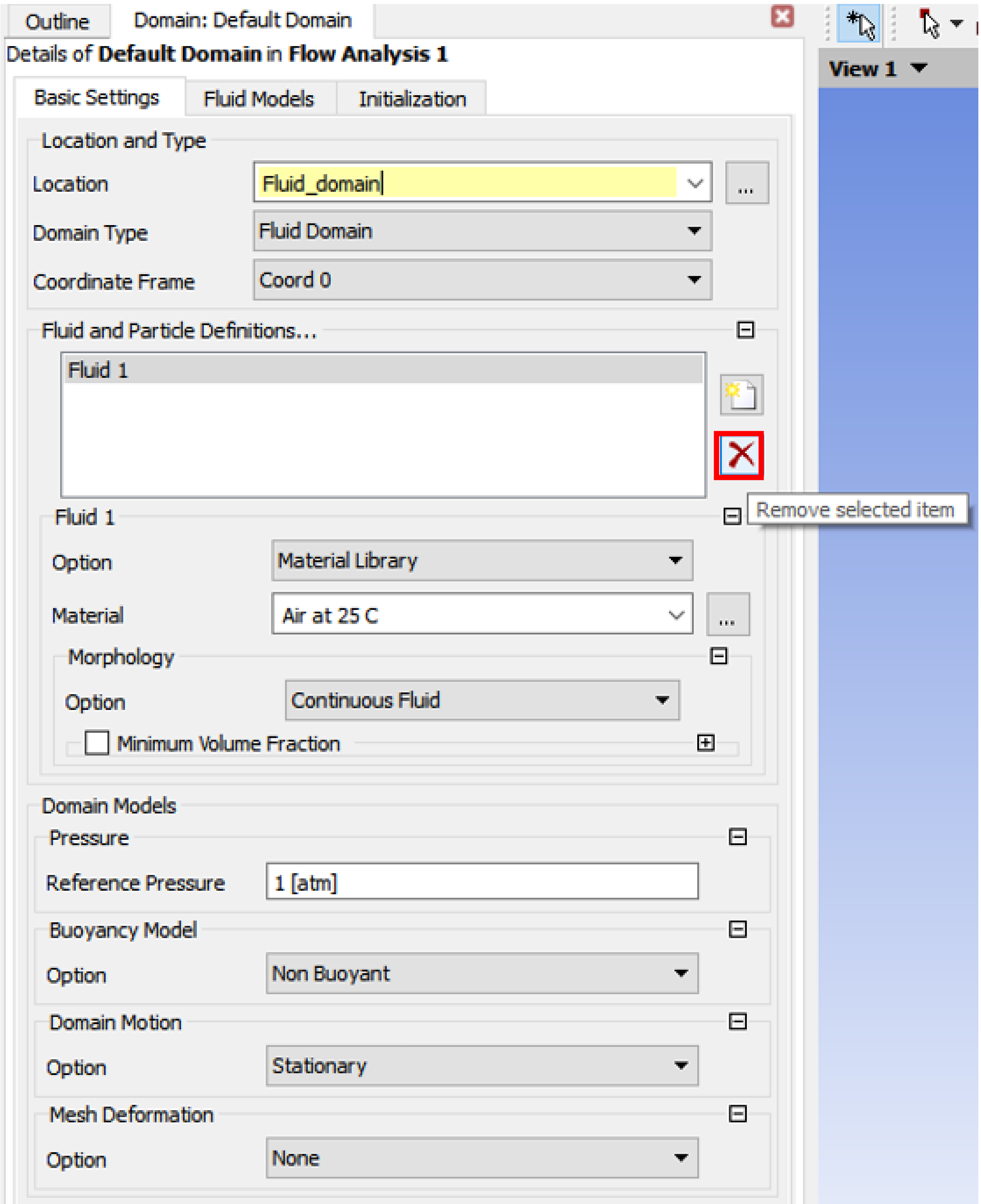
3) Double-click LMB *Default Domain*



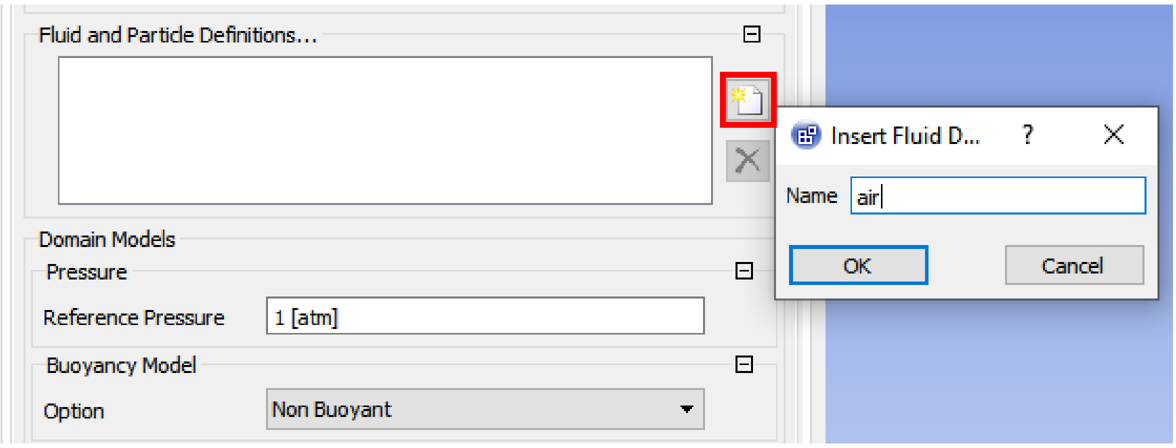
Apply the following and confirm *OK*.



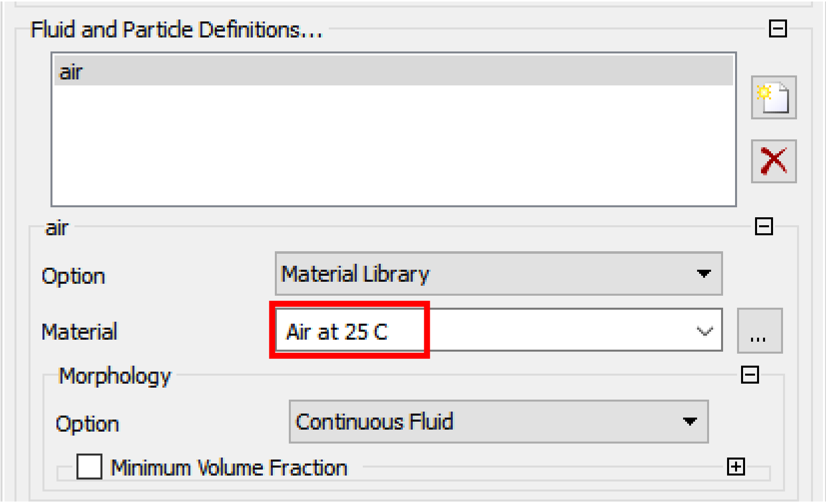
4) Delete default fluid *Fluid1*



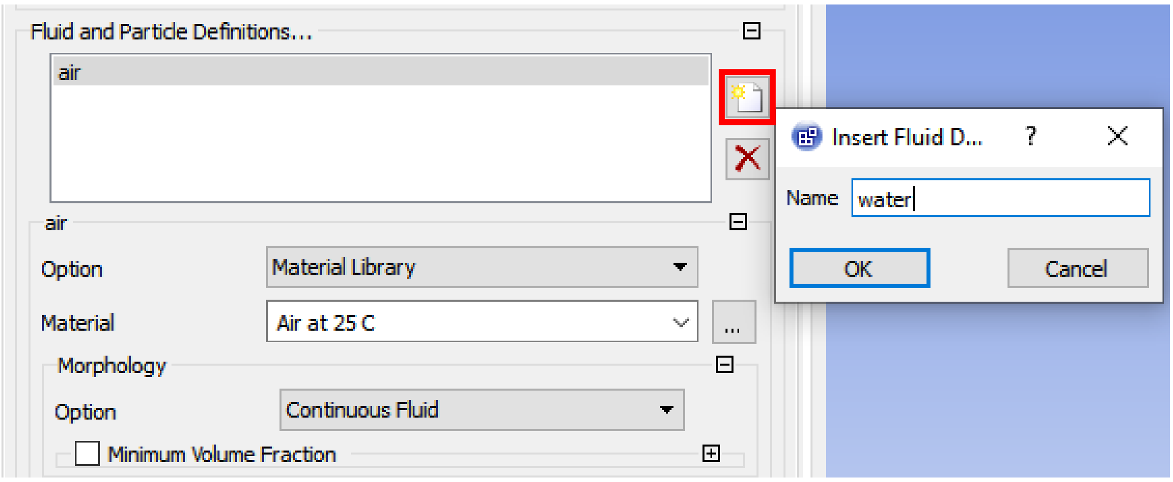
5) Add new fluid named *air*



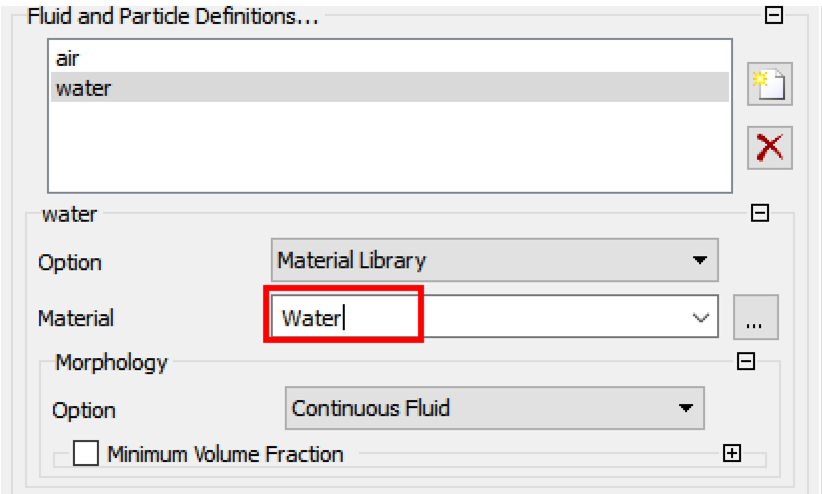
6) Select *Air at 25 C*



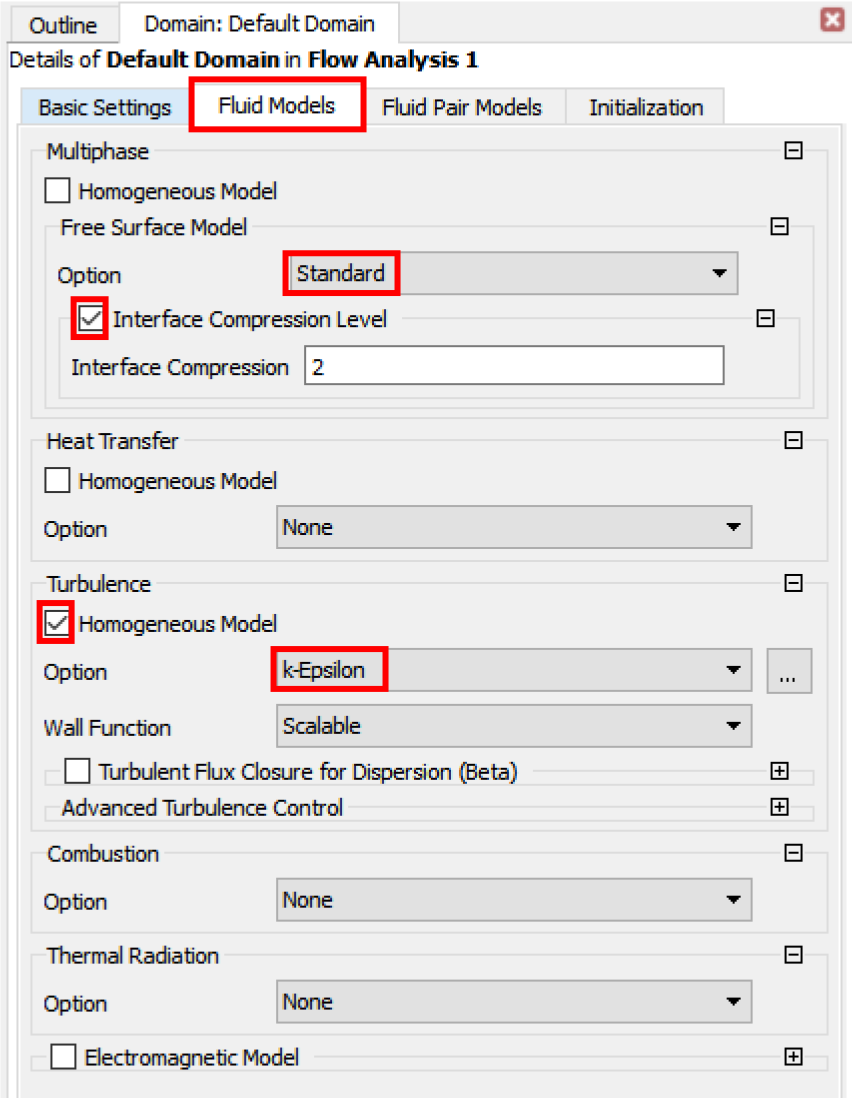
7) Create second fluid named *water*



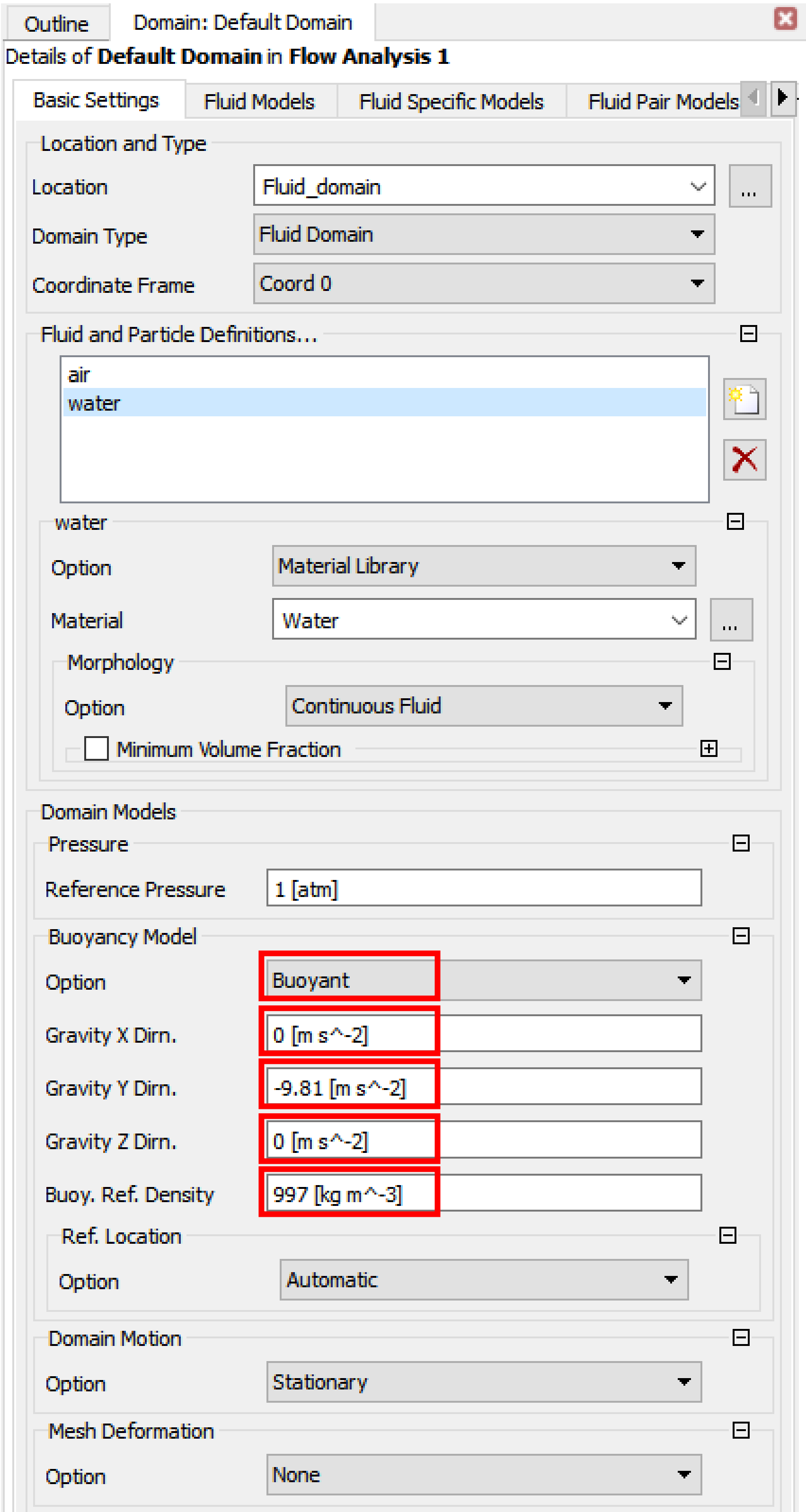
8) Select *Water*



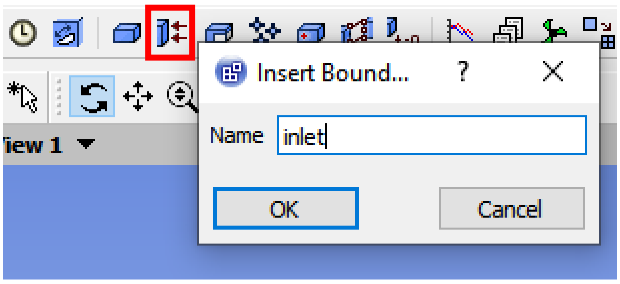
9) In tab *Fluid Models* apply the following



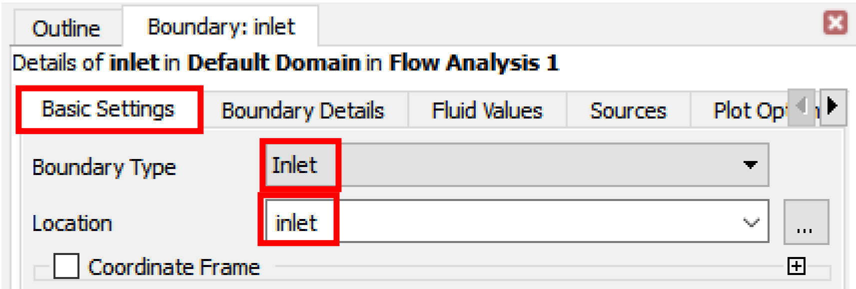
10) Go back to *Basic Settings* tab and apply below settings. Next confirm *OK*.

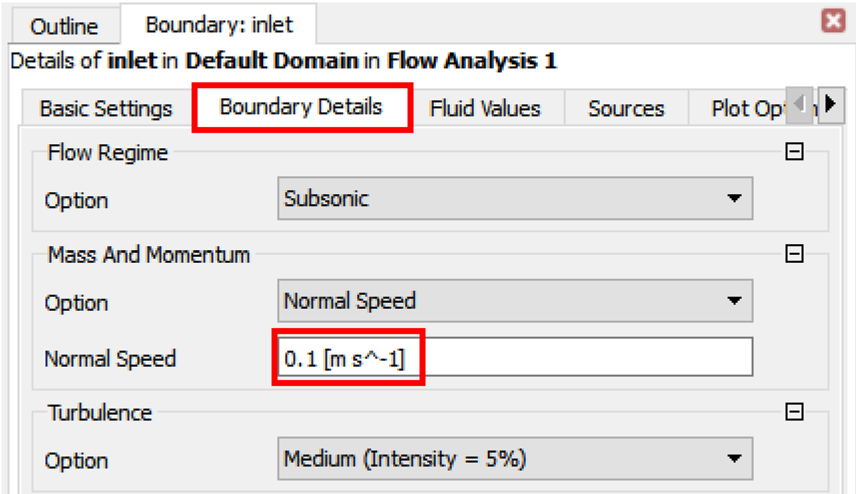


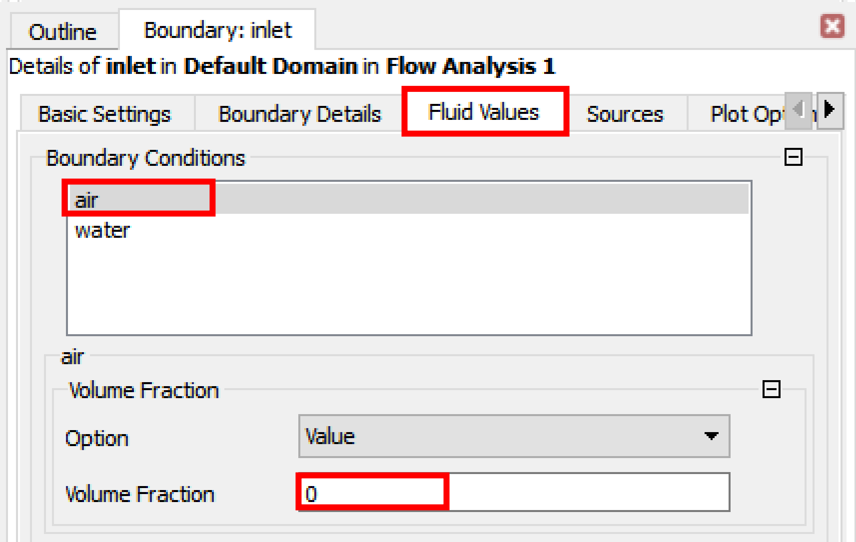
11) Create *inlet* boundary condition

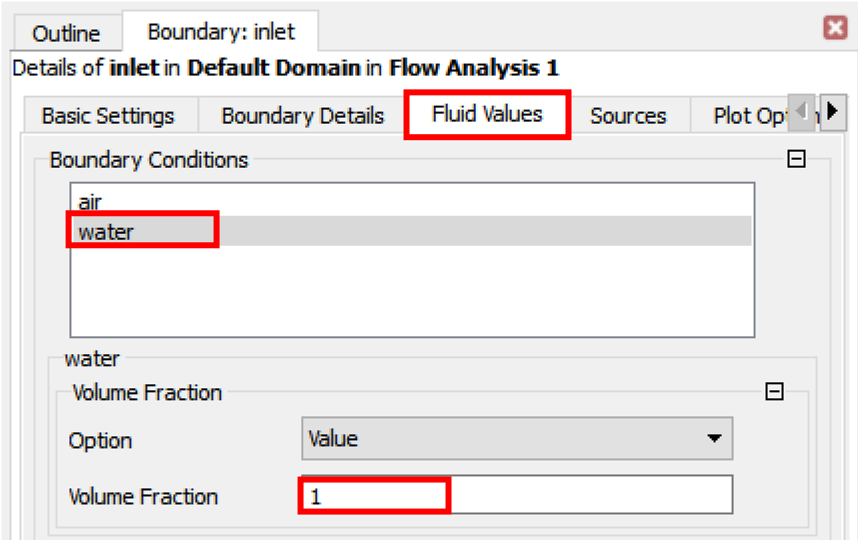


Apply as below and confirm *OK*.

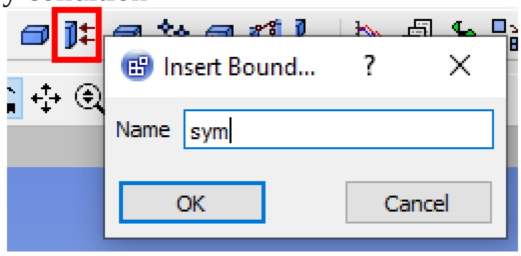




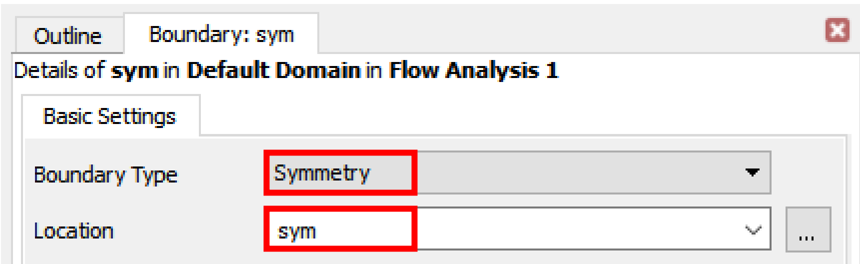




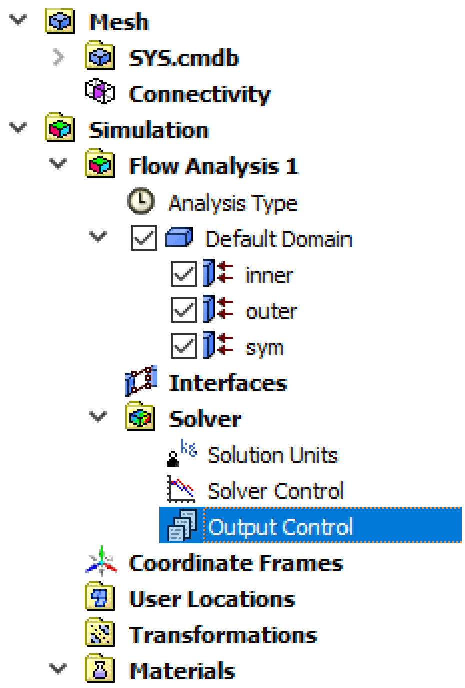
12) Create *sym* boundary condition



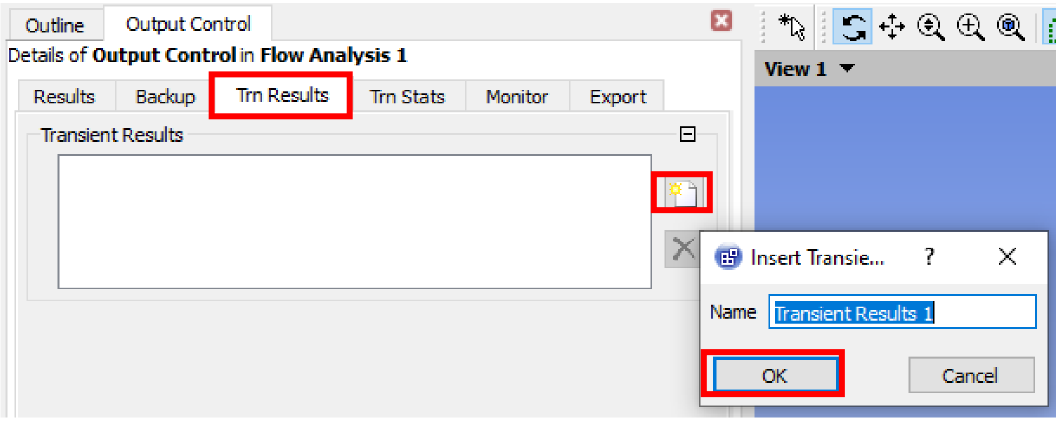
Apply as below and confirm *OK*.

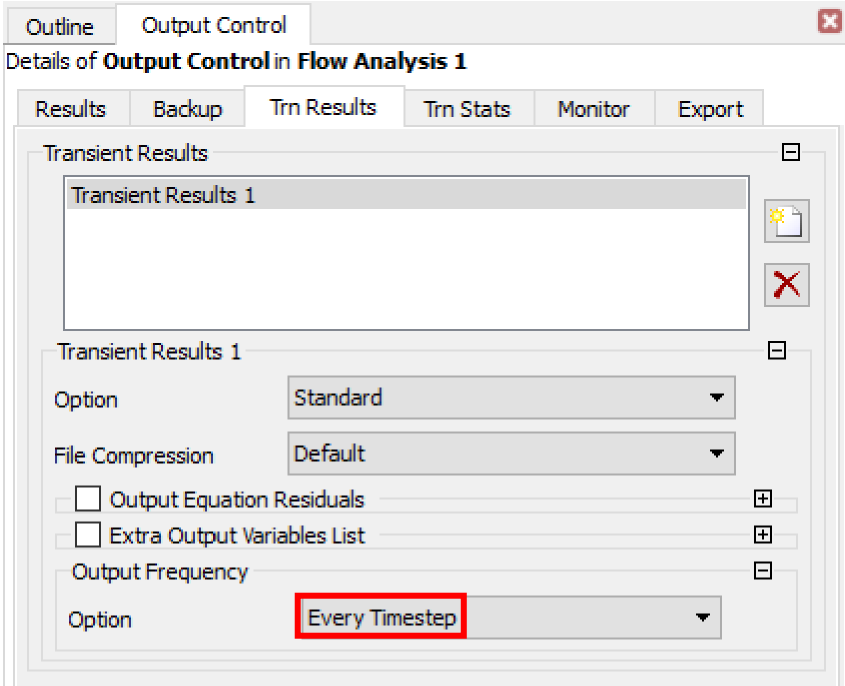


13) Open *Output Control*



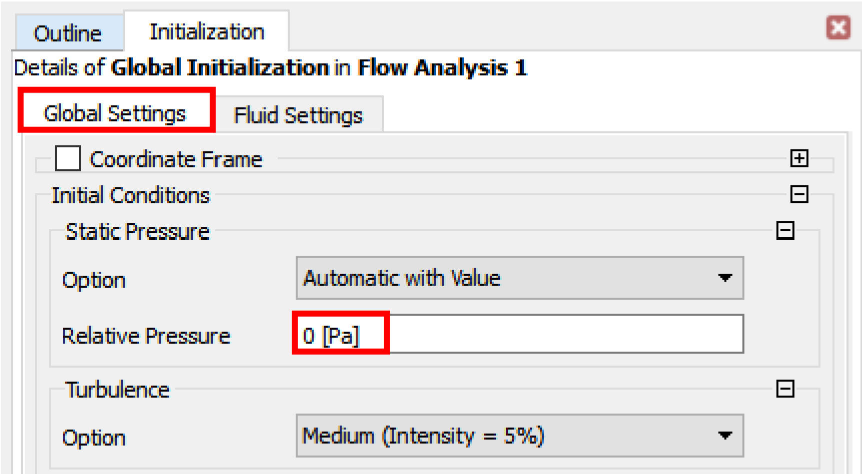
Apply as below and confirm *OK*.

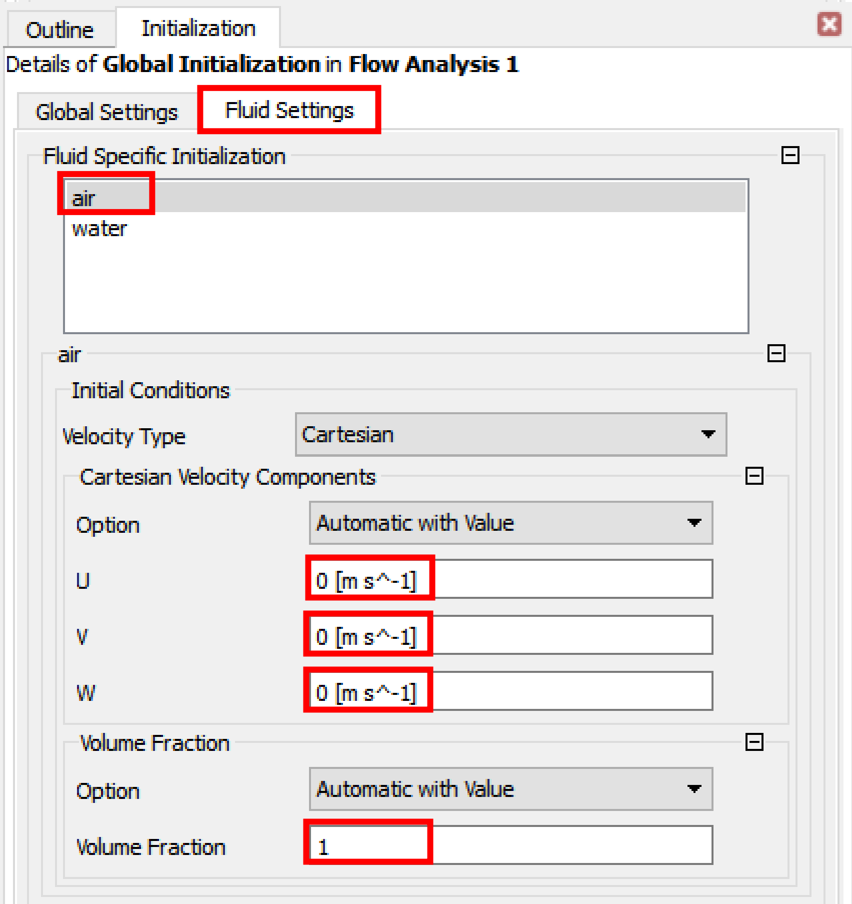


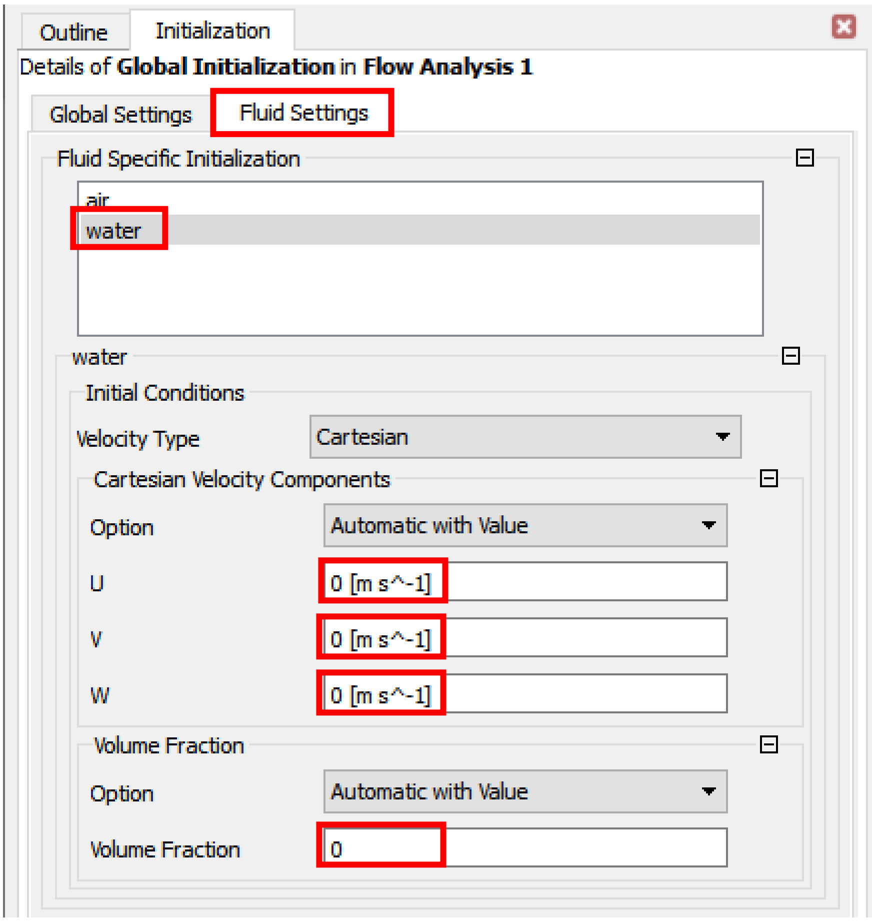


14) Create initial conditions by double-clicking LMB *Global Initialization* icon (at the top of the screen, almost in the middle)

Apply as below and confirm *OK*.


Outline
Initialization
Details of **Global Initialization** in **Flow Analysis 1**
Global Settings
Fluid Settings
Coordinate Frame
Initial Conditions
Static Pressure
Option
Automatic with Value
Relative Pressure
0 [Pa]
Turbulence
Option
Medium (Intensity = 5%)

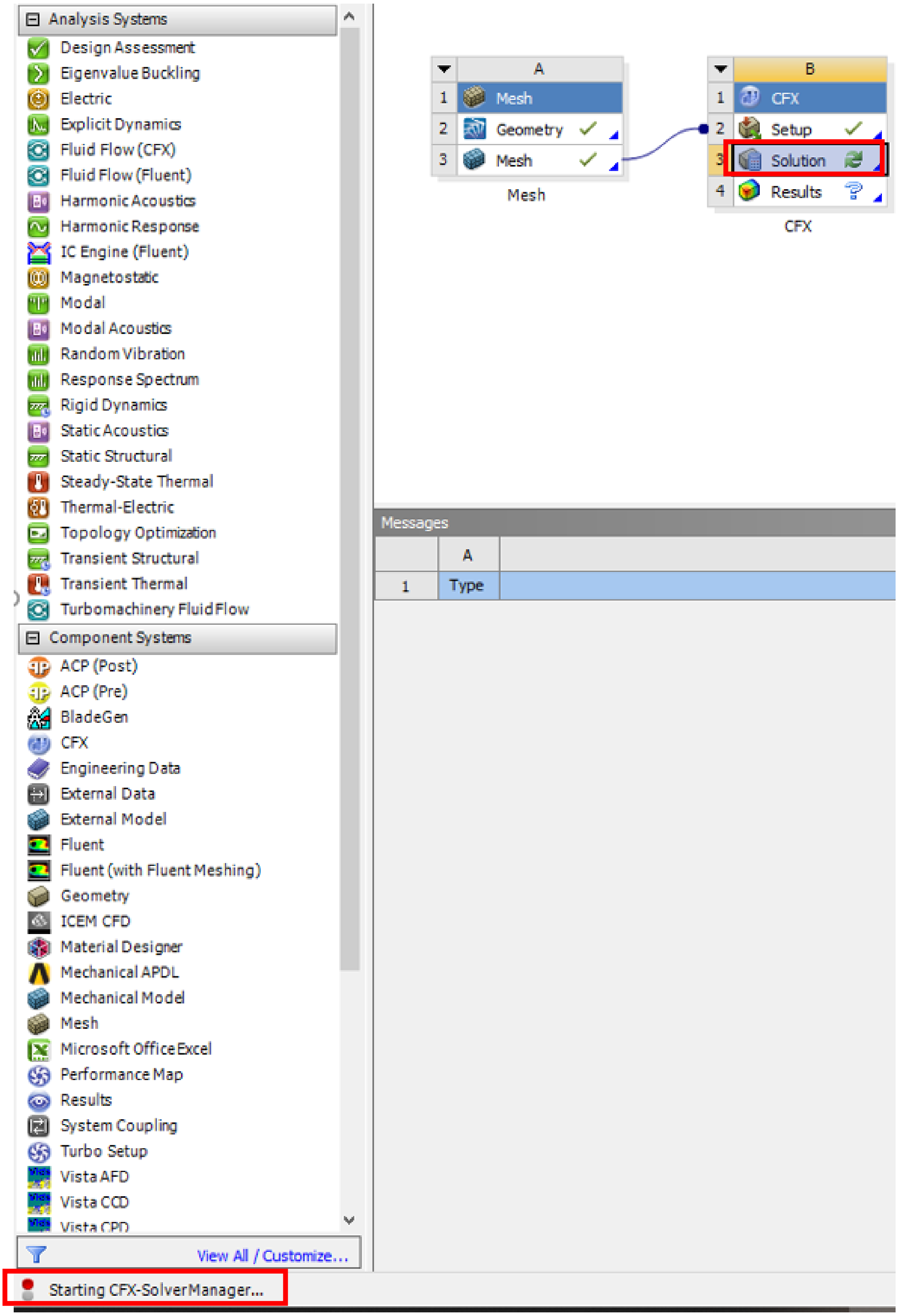

Outline
Initialization
Details of **Global Initialization** in **Flow Analysis 1**
Global Settings
Fluid Settings
Fluid Specific Initialization
air
water
air
Initial Conditions
Velocity Type
Cartesian
Cartesian Velocity Components
Option
Automatic with Value
U
0 [m s^-1]
V
0 [m s^-1]
W
0 [m s^-1]
Volume Fraction
Option
Automatic with Value
Volume Fraction
1



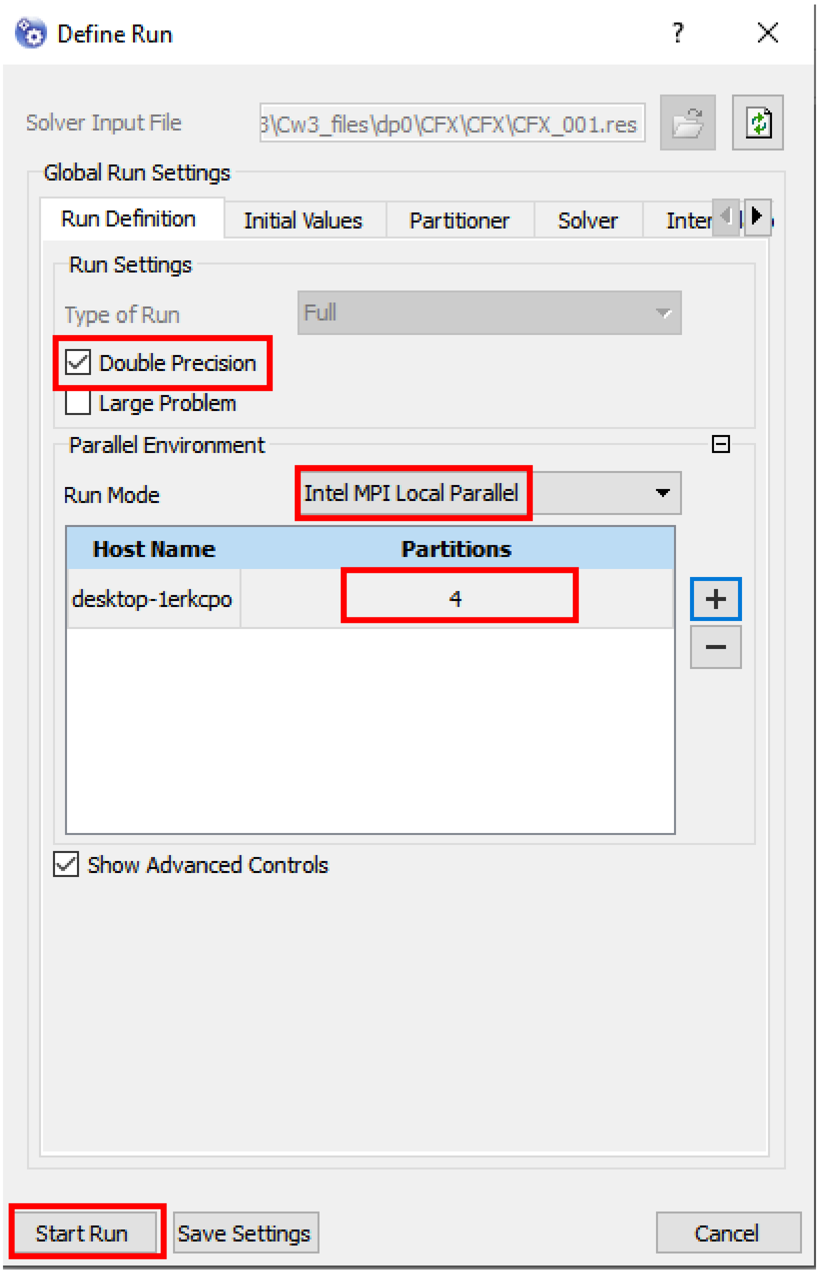
15) Close *Ansys CFX*.

2.4. Calculations

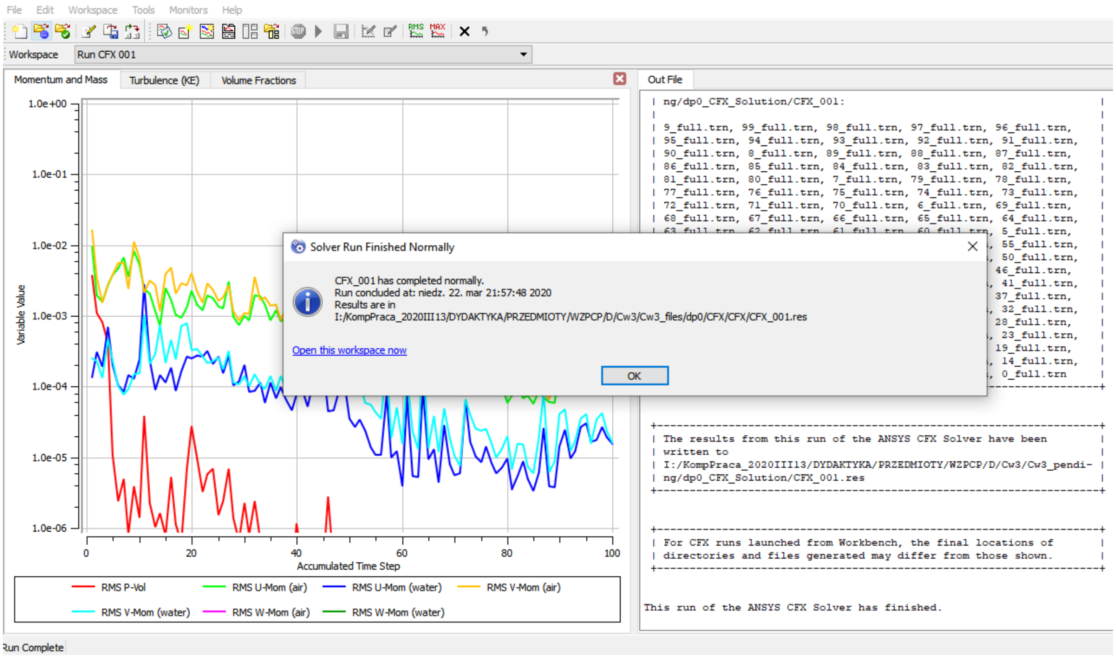
1) Double-click LMB on *Solution* to run *Ansys CFX Solver Manager*



2) Apply the following settings and press *Start Run*. The program will perform calculations. Wait a few moments for the message to complete the calculations.



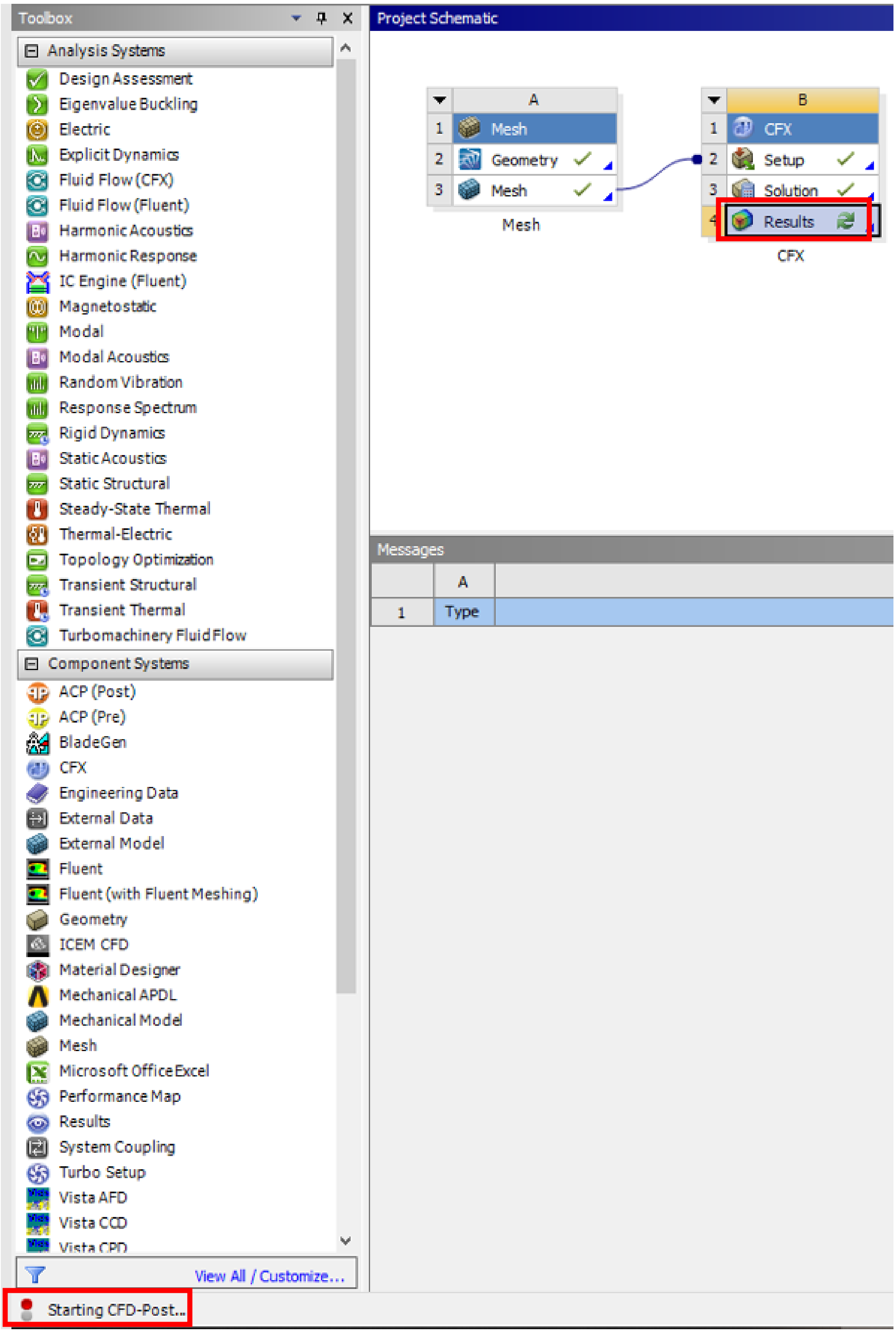
3) Calculations take about 15-20 minutes. After completing the calculations, the program will display a message:



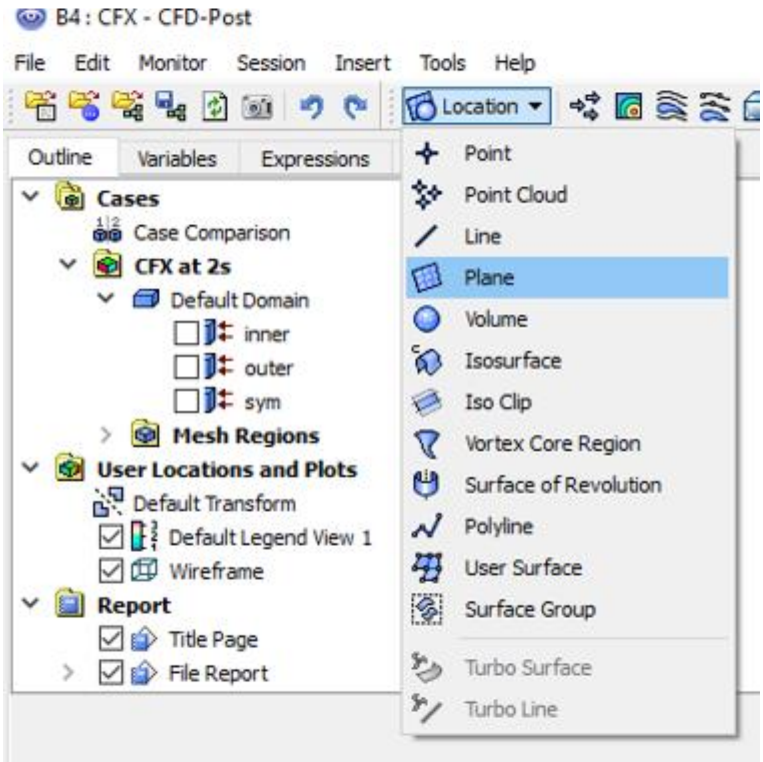
4) Confirm *OK* and close *Ansys CFX Solver Manager*. Save project in *Workbench.*

2.5. RESULTS PREPARATION

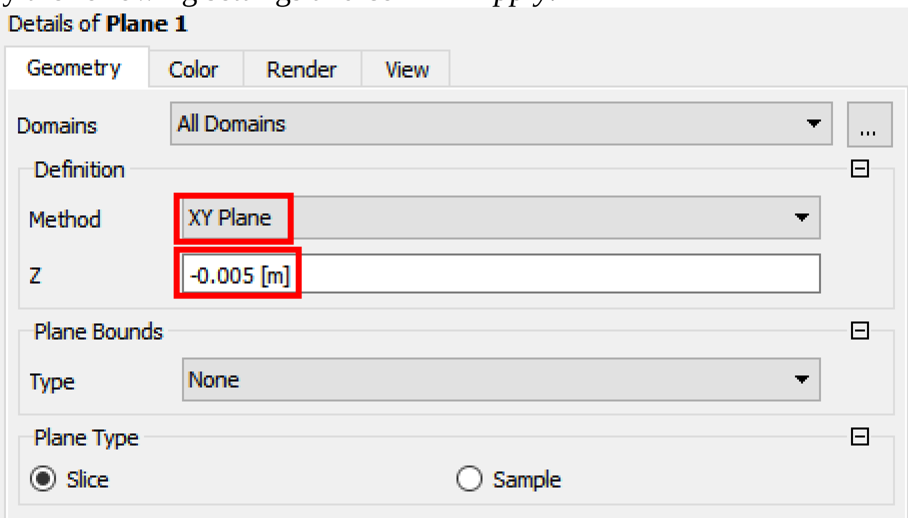
1) Double-click LMB on *Results* to run *Ansys CFD Post* and see the results



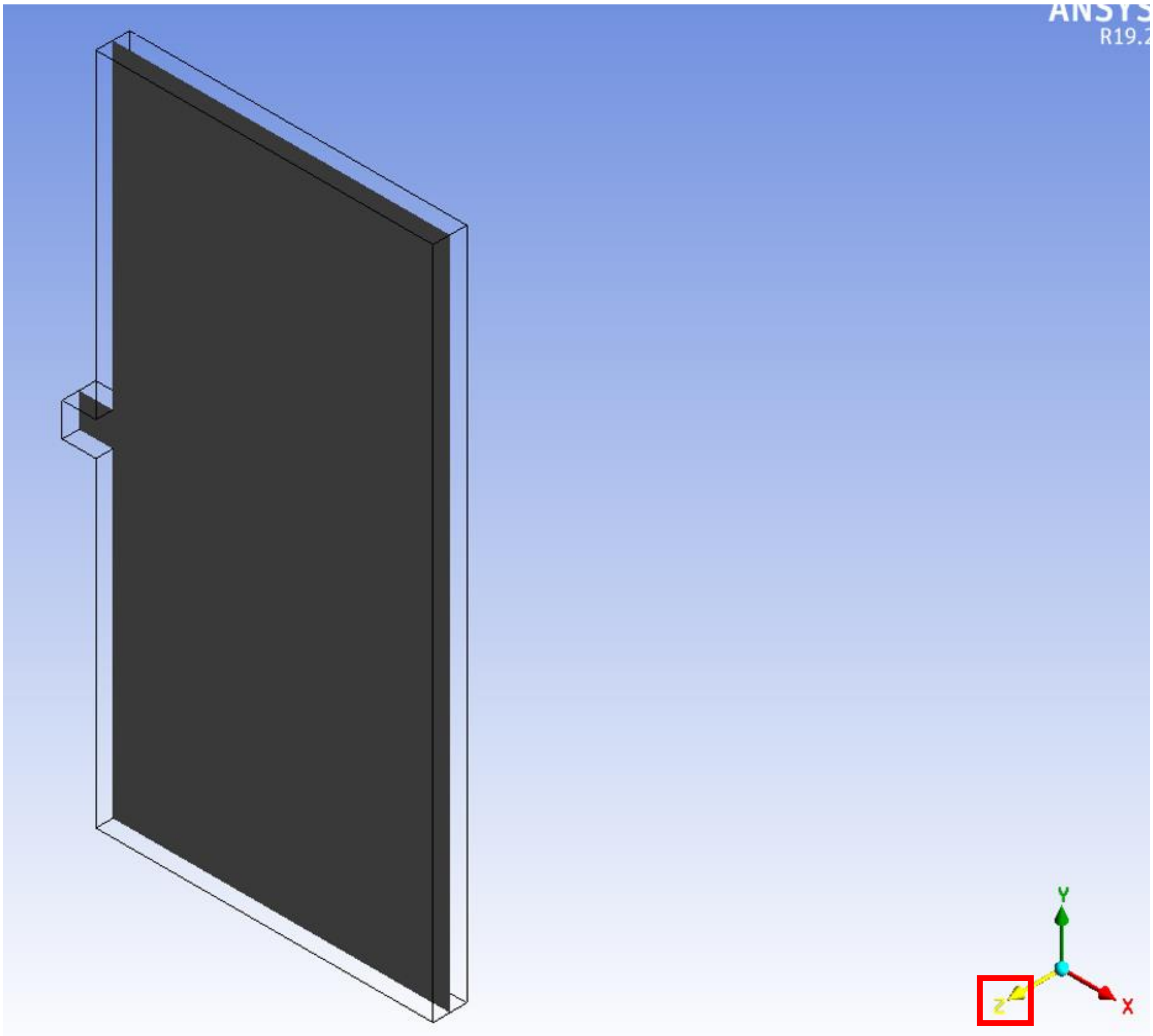
2) From menu *Location* select *Plane*



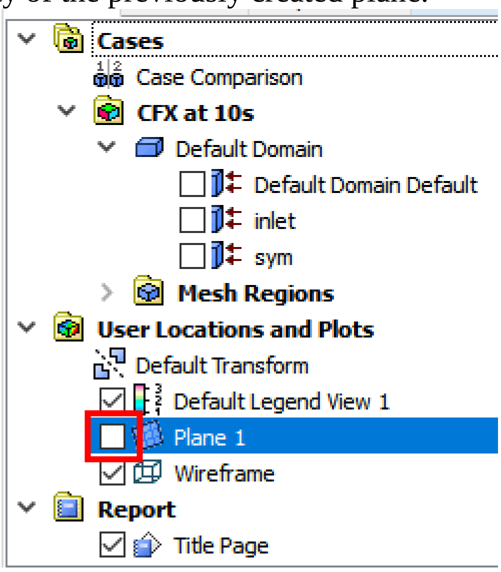
Apply the following settings and confirm *Apply*.



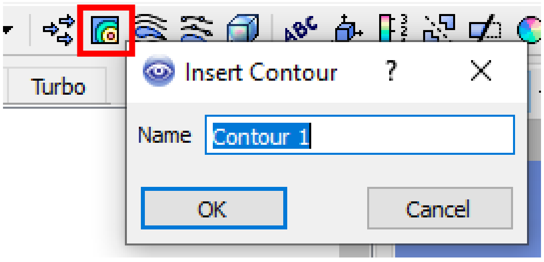
3) LMB click X axis



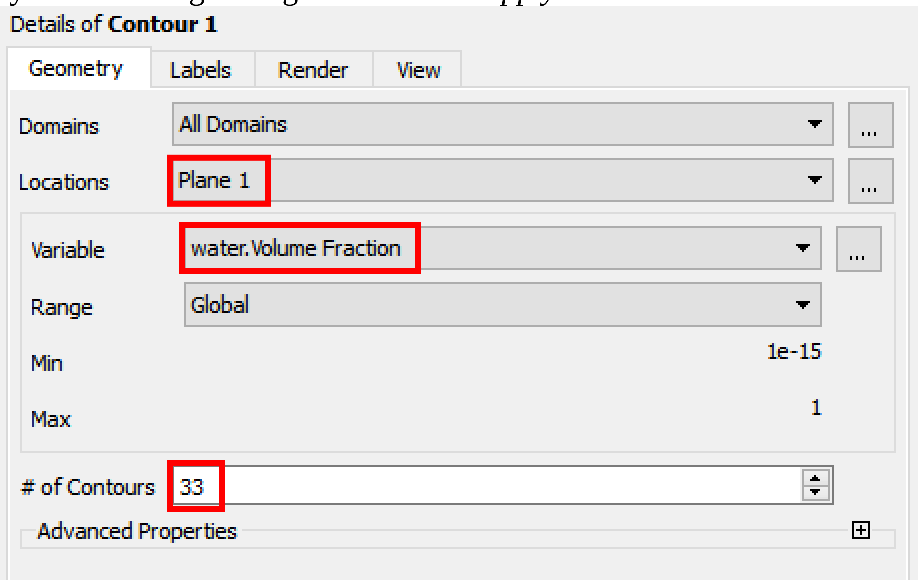
4) Uncheck the visibility of the previously created plane.



5) Select contour creation and confirm *OK*.

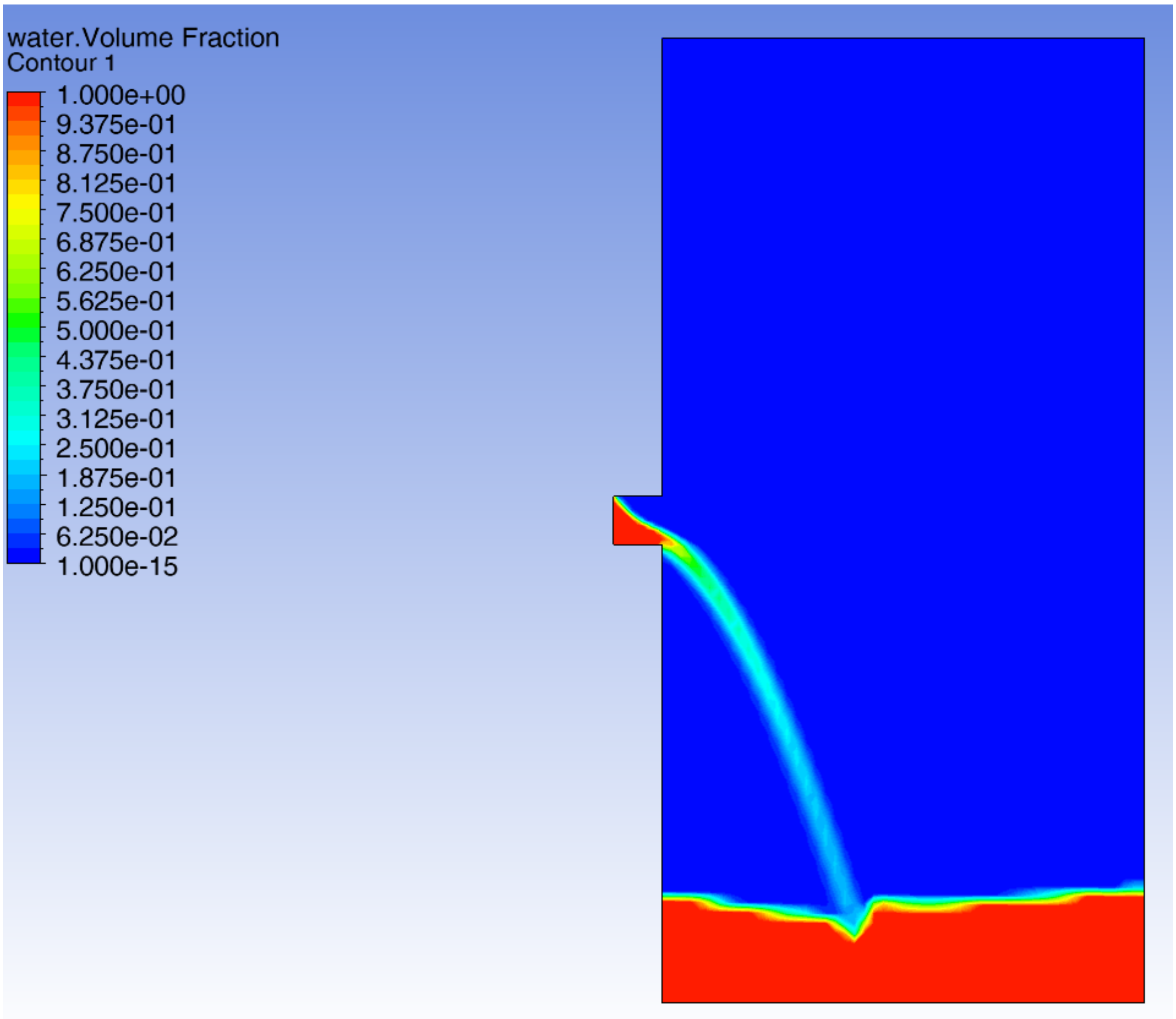


Apply the following settings and confirm *Apply*.

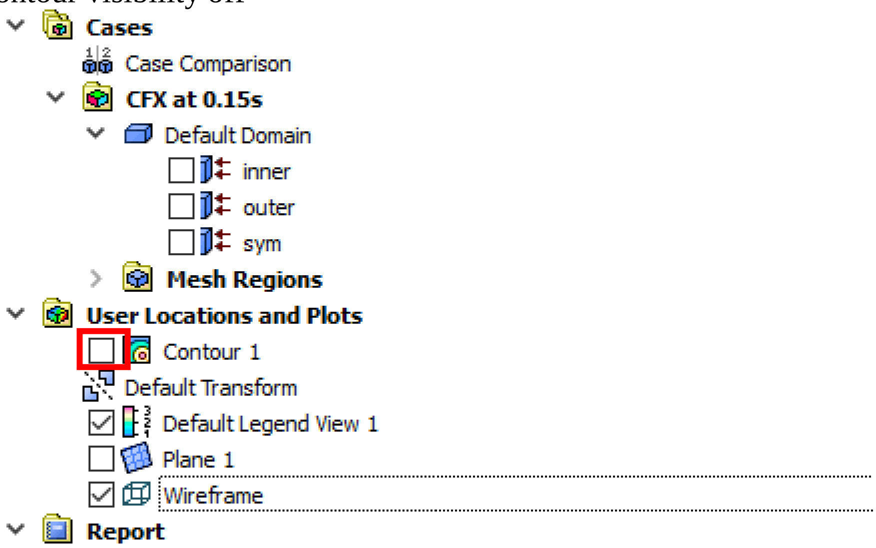


6) Select *Time Step Selector* icon

and by changing the simulation time, observe the changes in the volume of water in the tank over time



7) After saving contour pictures for the times 0; 1; 2; 3; 4; 5; 6; 7; 8; 9; 10 s. turn contour visibility off



8) Prepare and save pressure contours for times 0; 1; 2; 3; 4; 5; 6; 7; 8; 9; 10 s.
9) Additional task: Create an animation showing the changes in the volume of water in the tank (help: http://fluid.itcmp.pwr.wroc.pl/~pblasiak/CFD/UsefulInformation/animationCFX.jpg)

Results to be included in the report:

1) Contours of the distribution of the volume fraction of water in the tank for times: 0; 1; 2; 3; 4; 5; 6; 7; 8; 9; 10 s.
2) Contours of pressure distribution in the tank for times: 0; 1; 2; 3; 4; 5; 6; 7; 8; 9; 10 s.