Politechnika Wrocławska

Mechanical and Power Engineering Faculty
Full-time studies

Selected problems of thermal-flow processes

Exercise no. 5

Modeling of flow containing solid particles

Wrocław 2020

TABLE OF CONTENTS

1. INTRODUCTION

The exercise will show how to model fluid flow containing solid particles. The issue will be presented on the example of a simple cyclone. A mixture of air and wood particles at a speed of 1 m/s flows into the rectangular cyclone inlet. The wood particles are described by a normal distribution with an average particle diameter of 0.5 mm. The mixture of particles and air flows tangentially to the cylindrical part of the cyclone. Due to the centrifugal force, the dust particles move near the walls in a spiral motion and flow out at the bottom of the cyclone through outlet 1. Air cleaned from particles flows out through the upper outlet 2. The flow of the mixture is without heat exchange with the environment. The diagram of the analyzed case is presented in Fig. 1.

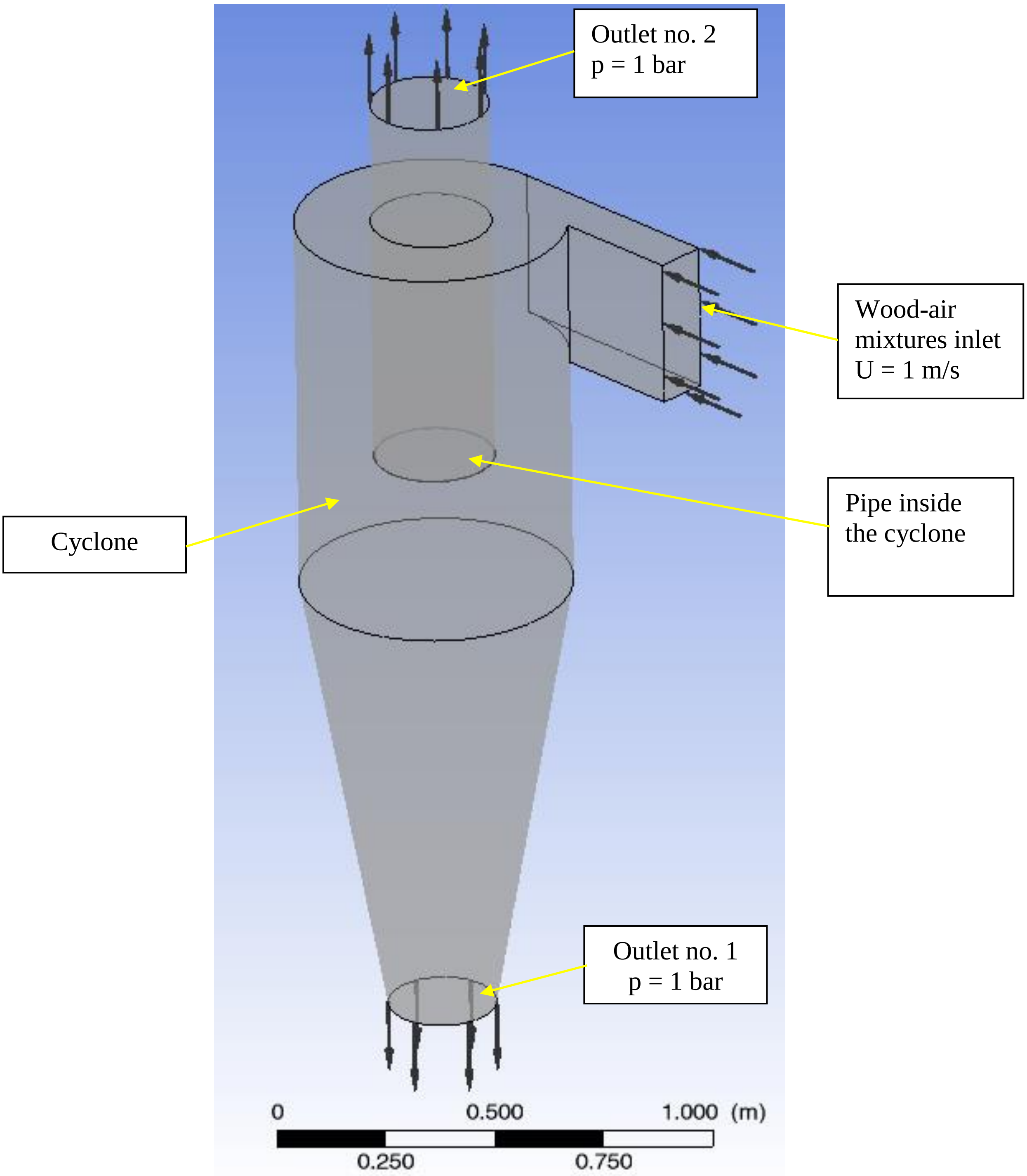


Fig. 1. Diagram of the issue of air flow containing wood particles in a cyclone

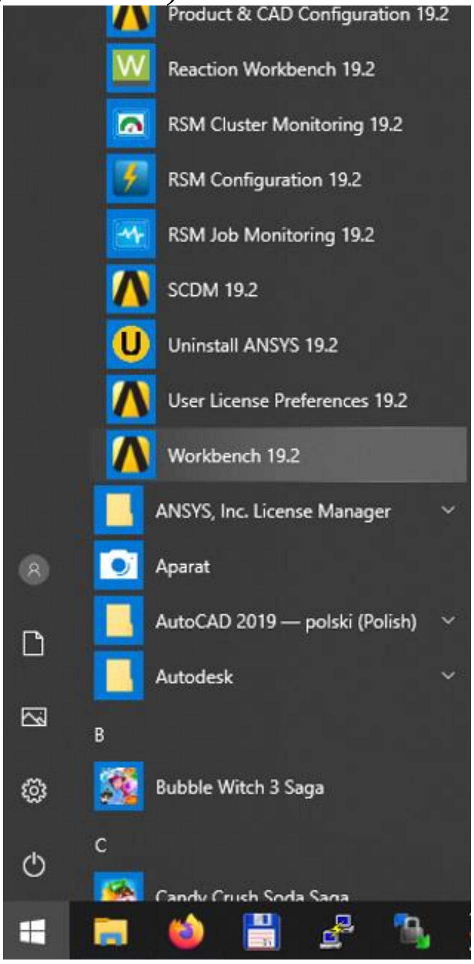
2. Flow through a cyclone

2.1. Geometry

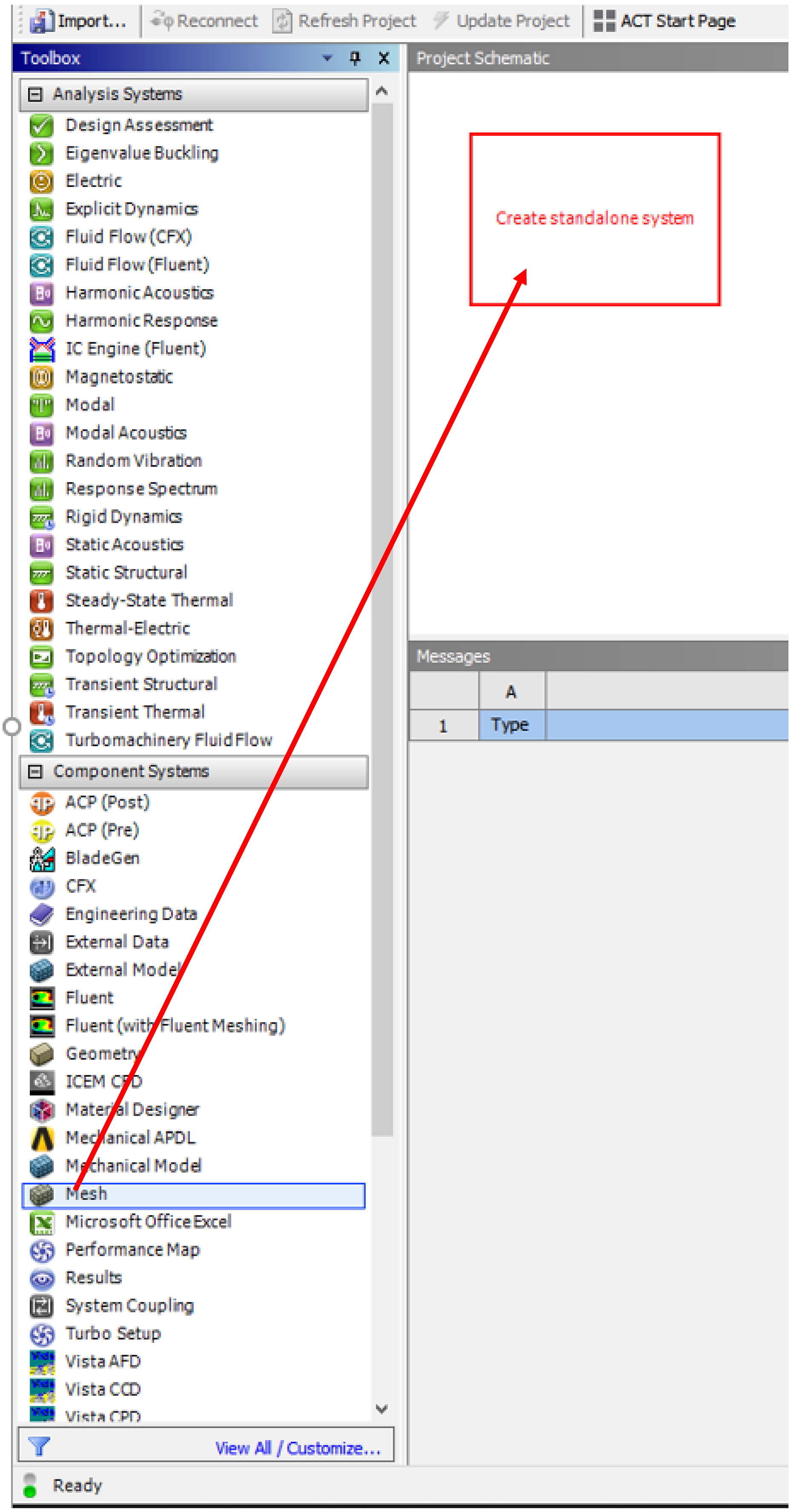
Do the following tasks:

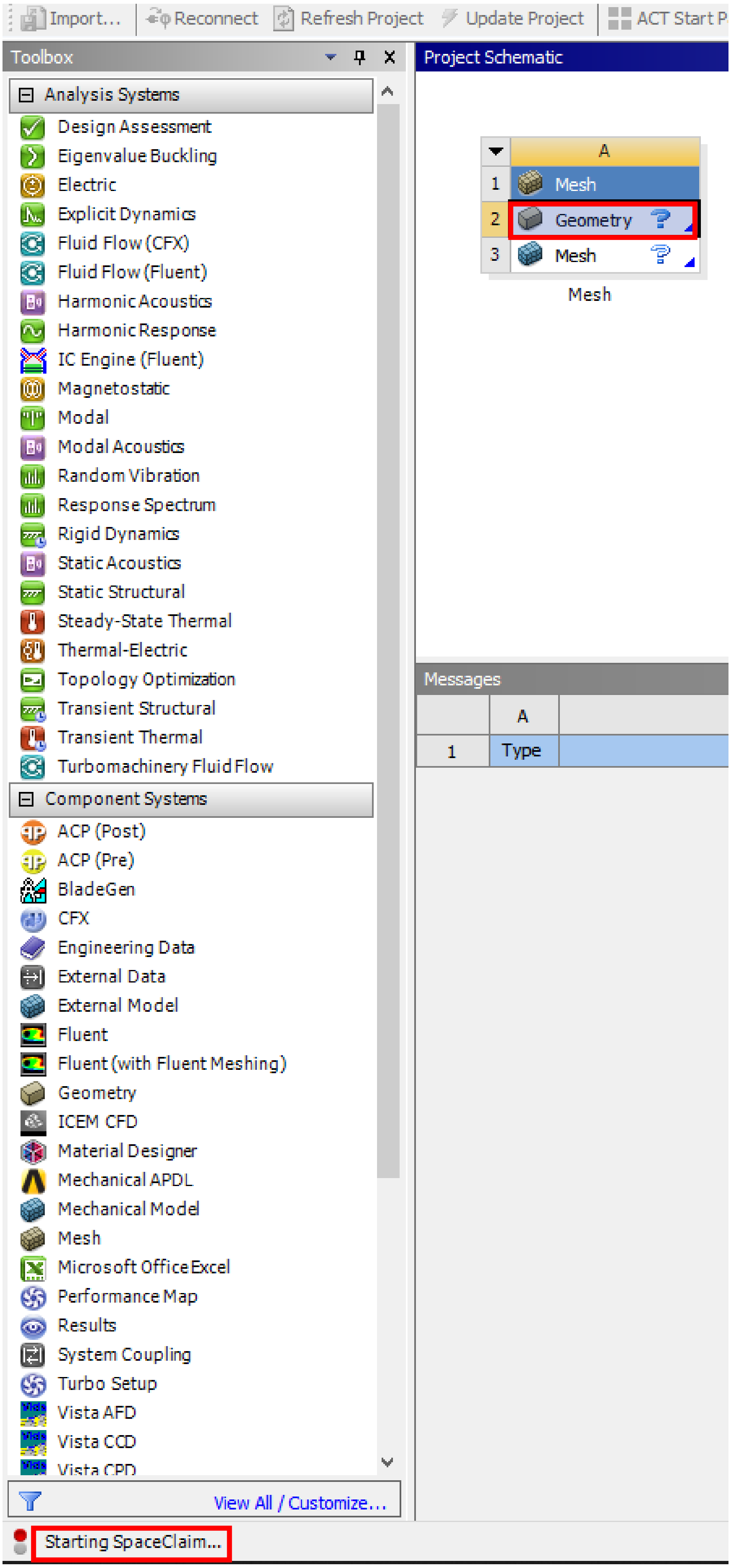
1) Open *Ansys Workbench* and save project as Ex5 in catalog Ex5 (*File->Save As*).
   RULE OF THUMB NO. 1: *We create a separate catalog for each project*

RULE OF THUMB NO. 2: *In catalog names we do not use: spaces, special characters (e.g. @ # $% ^ & * etc.) and Polish characters*



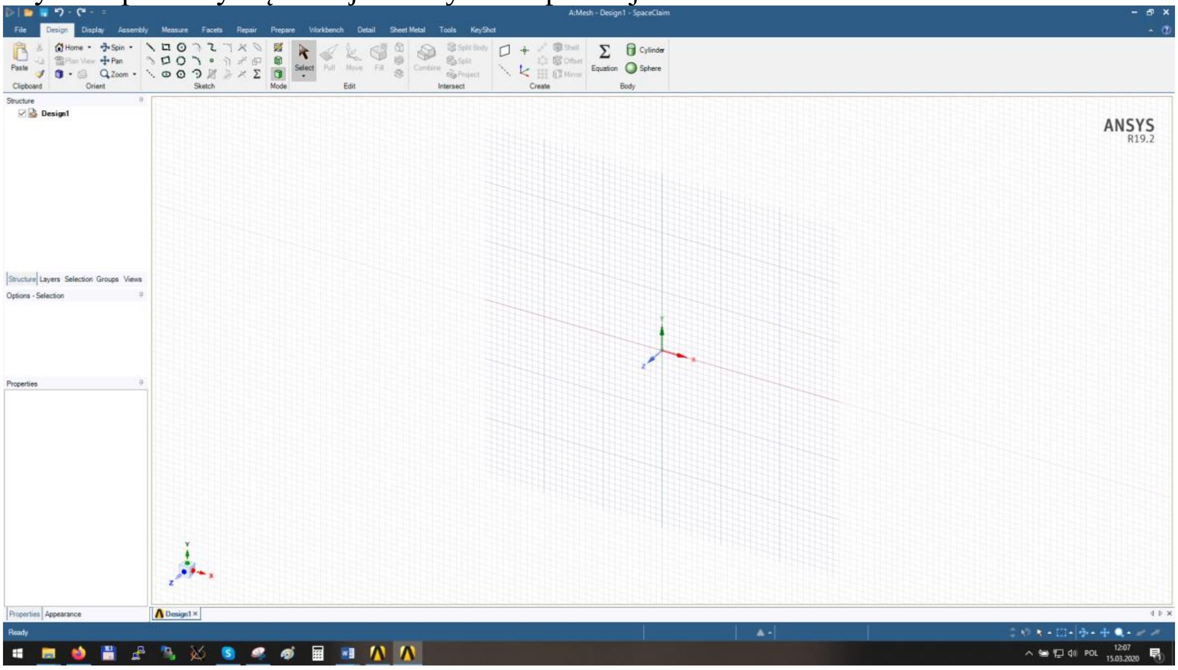
2) Select *Mesh* module and open *Spaceclaim.* To do this, hold the left mouse button (LMB) on the *Mesh* module and drag it to the *Project Schematic* field. Then double-click LMB on *Geometry* to start the *Spaceclaim* program in which the geometry will be created. Note that in the lower left corner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

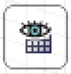

Import...
Reconnect
Refresh Project
Update Project
ACT Start Page
Toolbox
Project Schematic
Create standalone system
Analysis Systems
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (CFX)
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic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

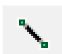

Import...
Reconnect
Refresh Project
Update Project
ACT Start P
Toolbox
Project Schematic
Analysis Systems
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (CFX)
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
Random 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...
A
1 Mesh
2 Geometry
3 Mesh
Mesh
Messages
A
1 Type
Starting SpaceClaim...

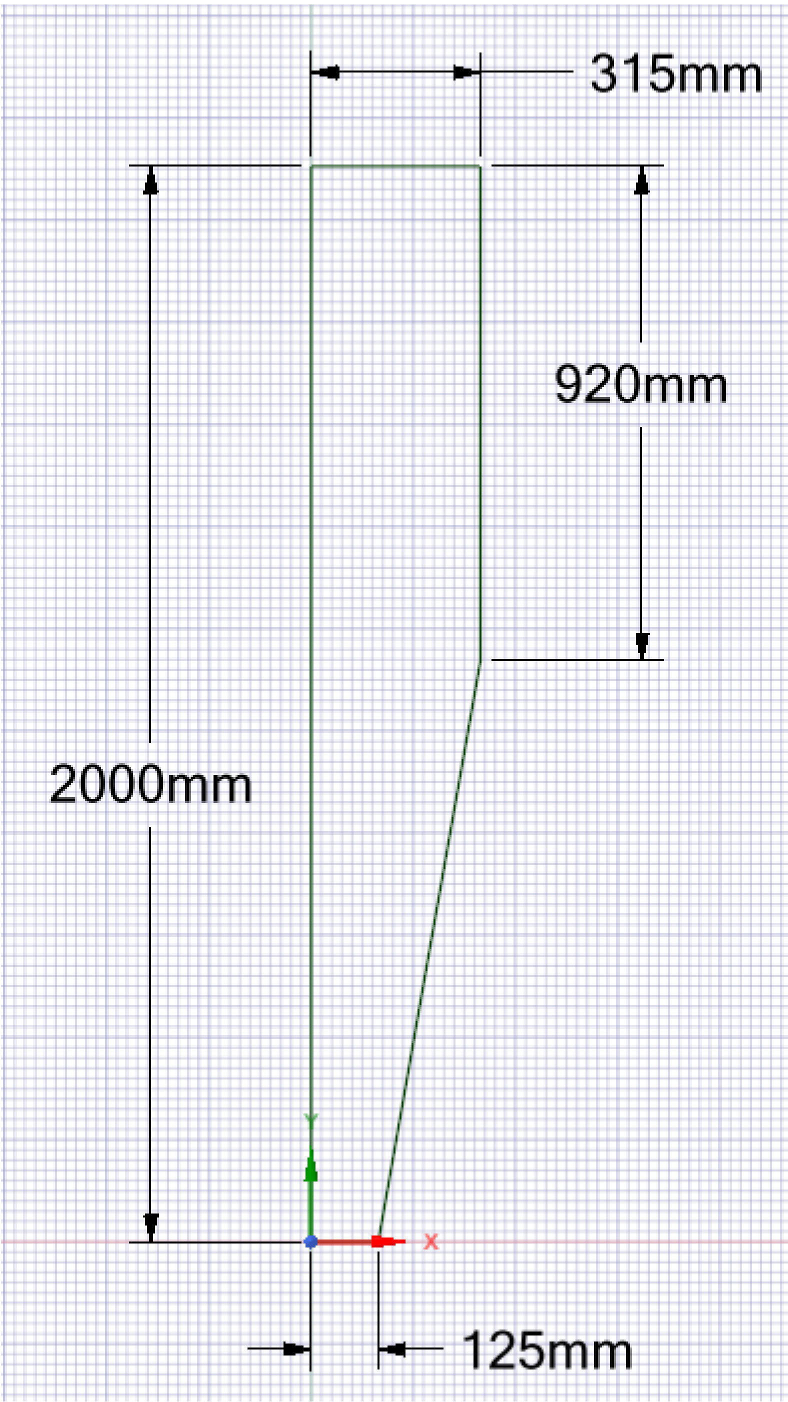
3) Click LMB *Select New Sketch* 

Wybierz płaszczyznę X-Y jak na rysunku poniżej.



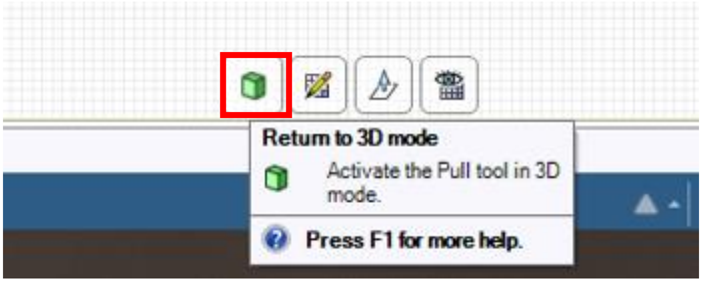
4) Click *Plan View*  to rotate the drawing plane parallel to the screen (you can also do this by pressing *Shift* + *v*).

5) In the panel at the top of the screen, select the line drawing icon  and draw a profile with dimensions as shown below.

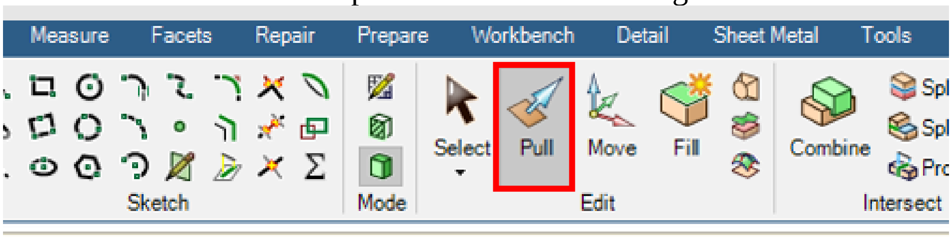


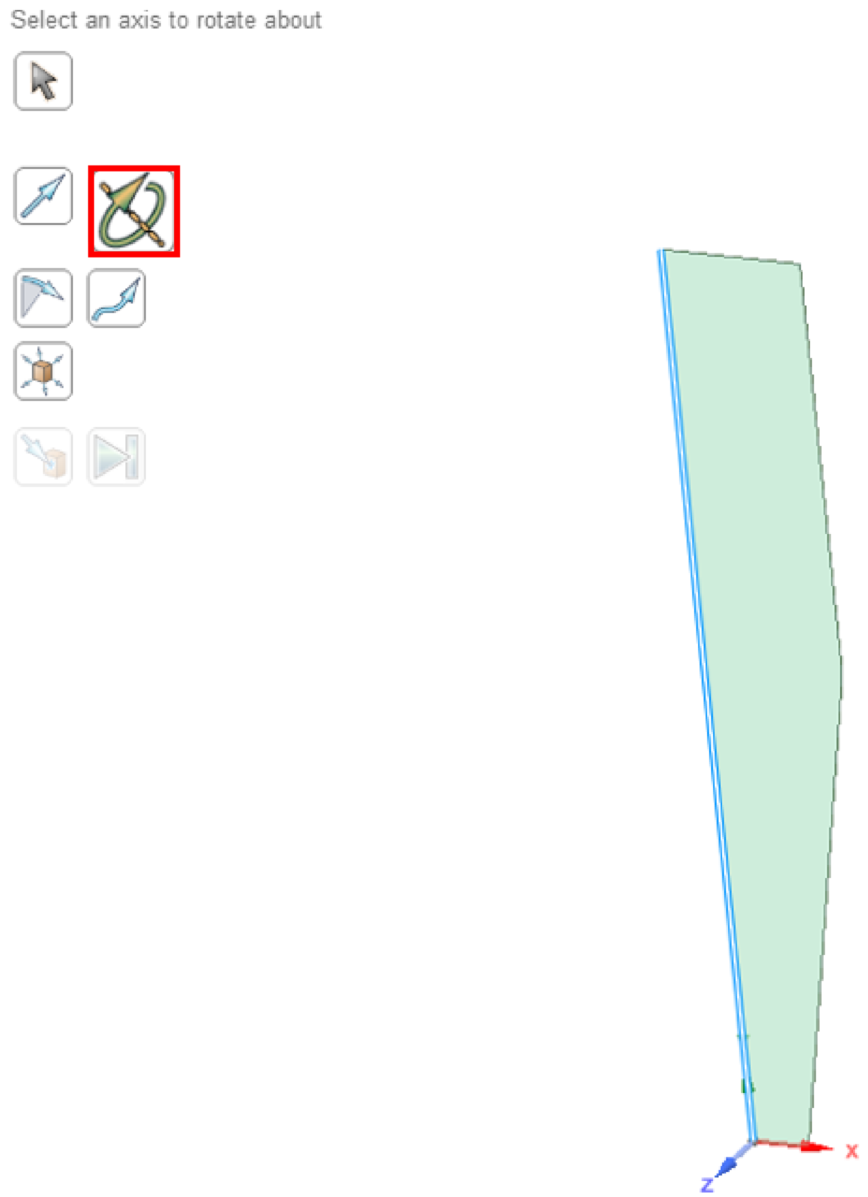
Remember that if you fail at any time, you can click the undo icon  (located in the upper left corner of the screen) or *Ctrl + z*.

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

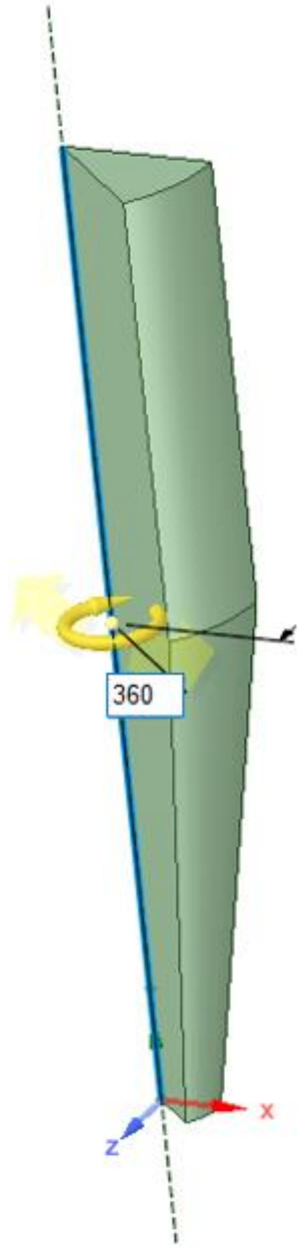


7) Select *Pull* and then *Revolve* option. Next select the long vertical line.

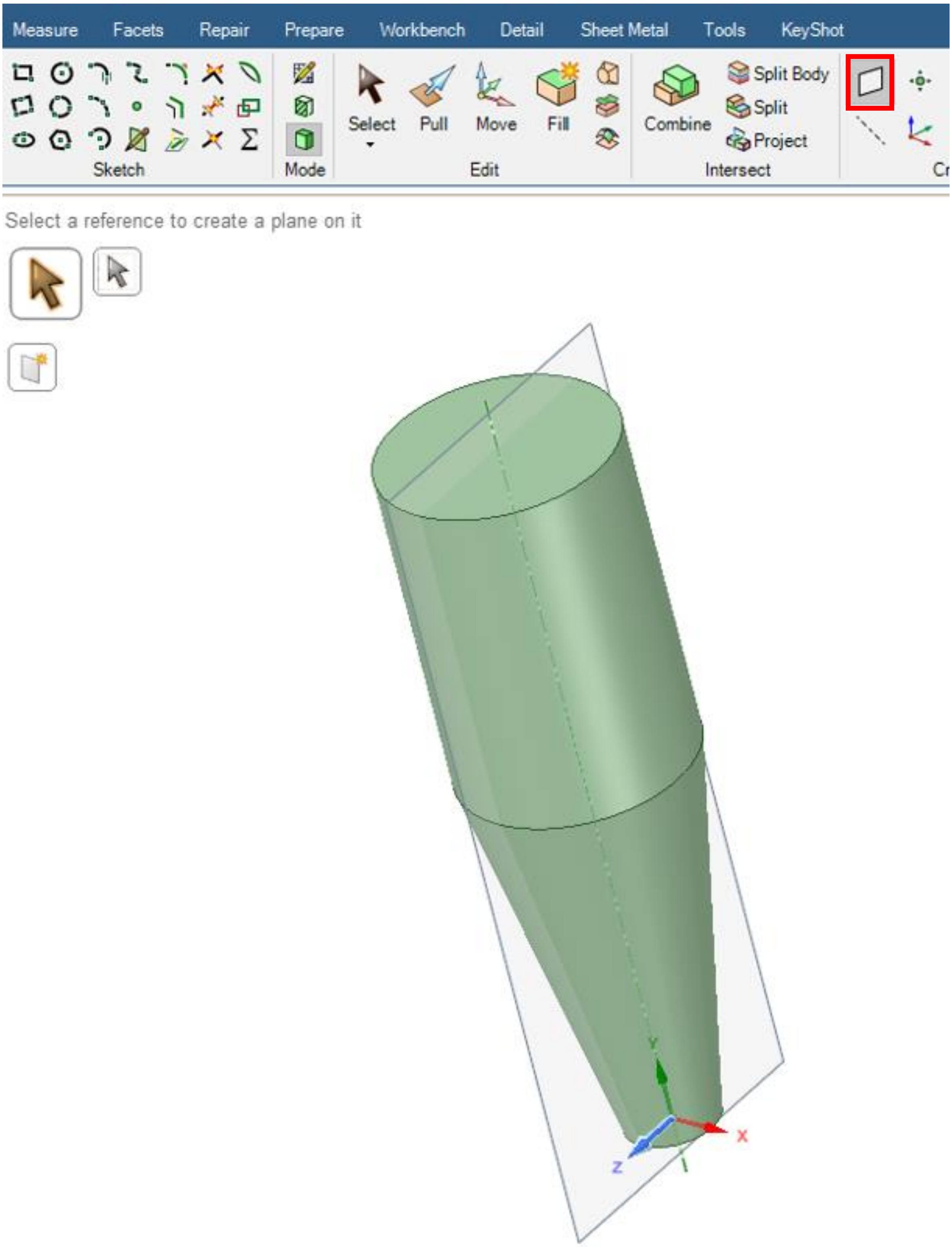




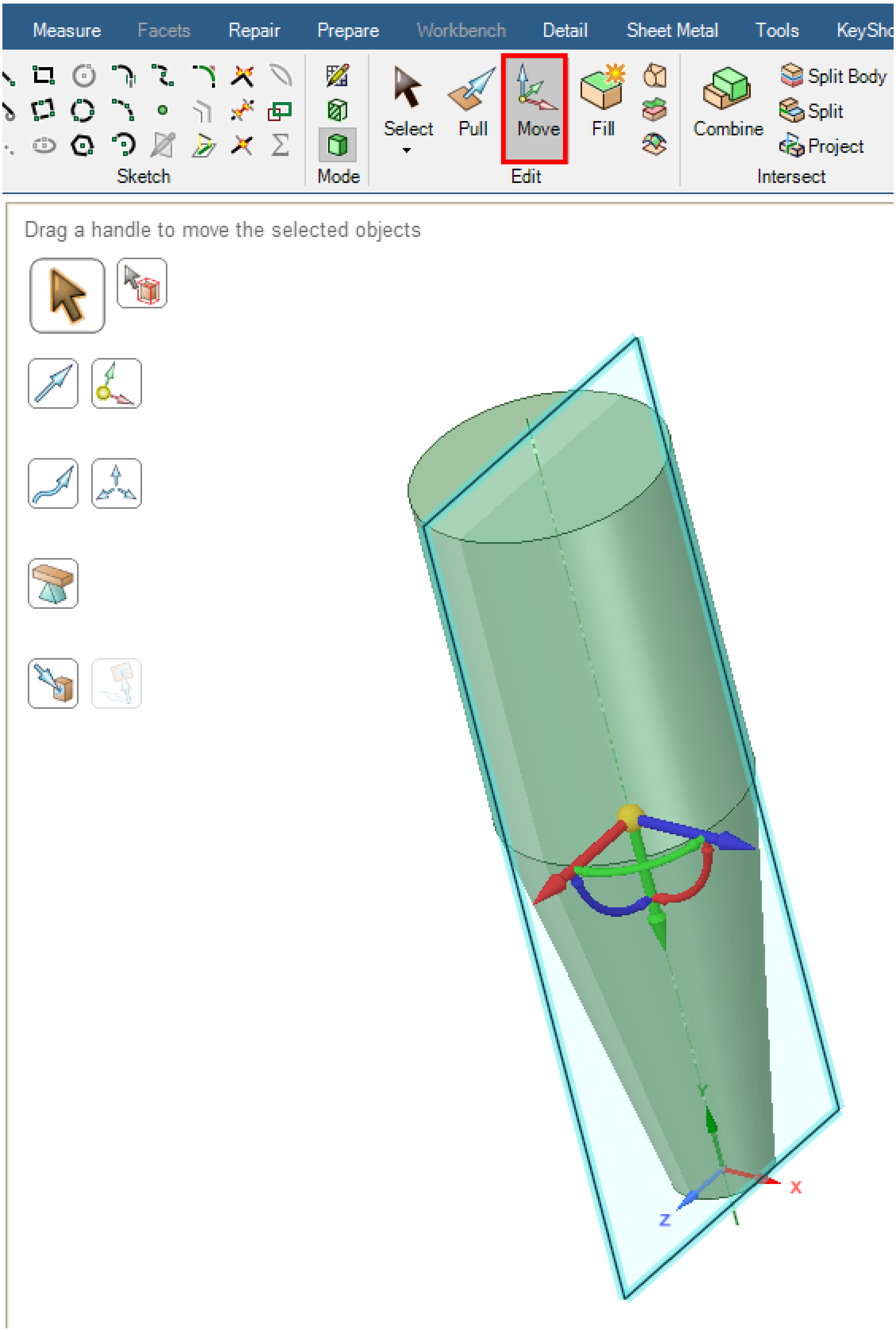
8) LMB click on the bent yellow arrow and holding the LMB move the mouse. Then, holding LMB, enter the value 360 and confirm *Enter*.



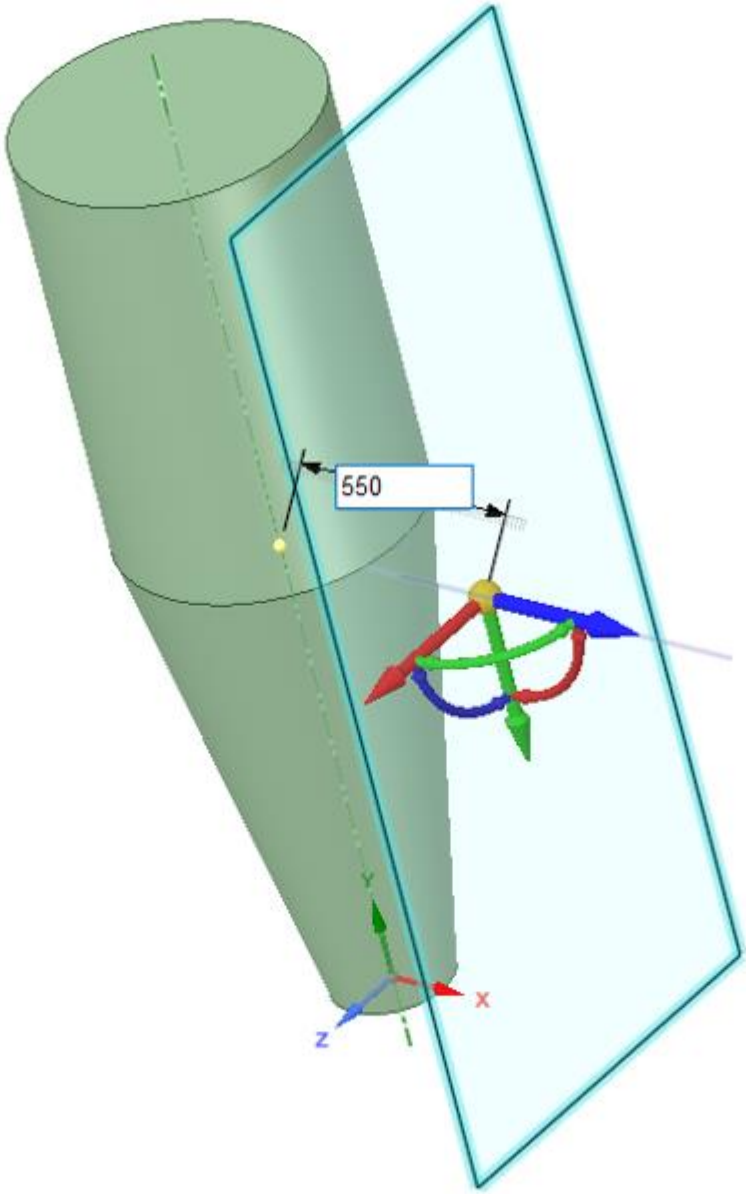
9) Select the plane creation icon and LMB select the Z axis



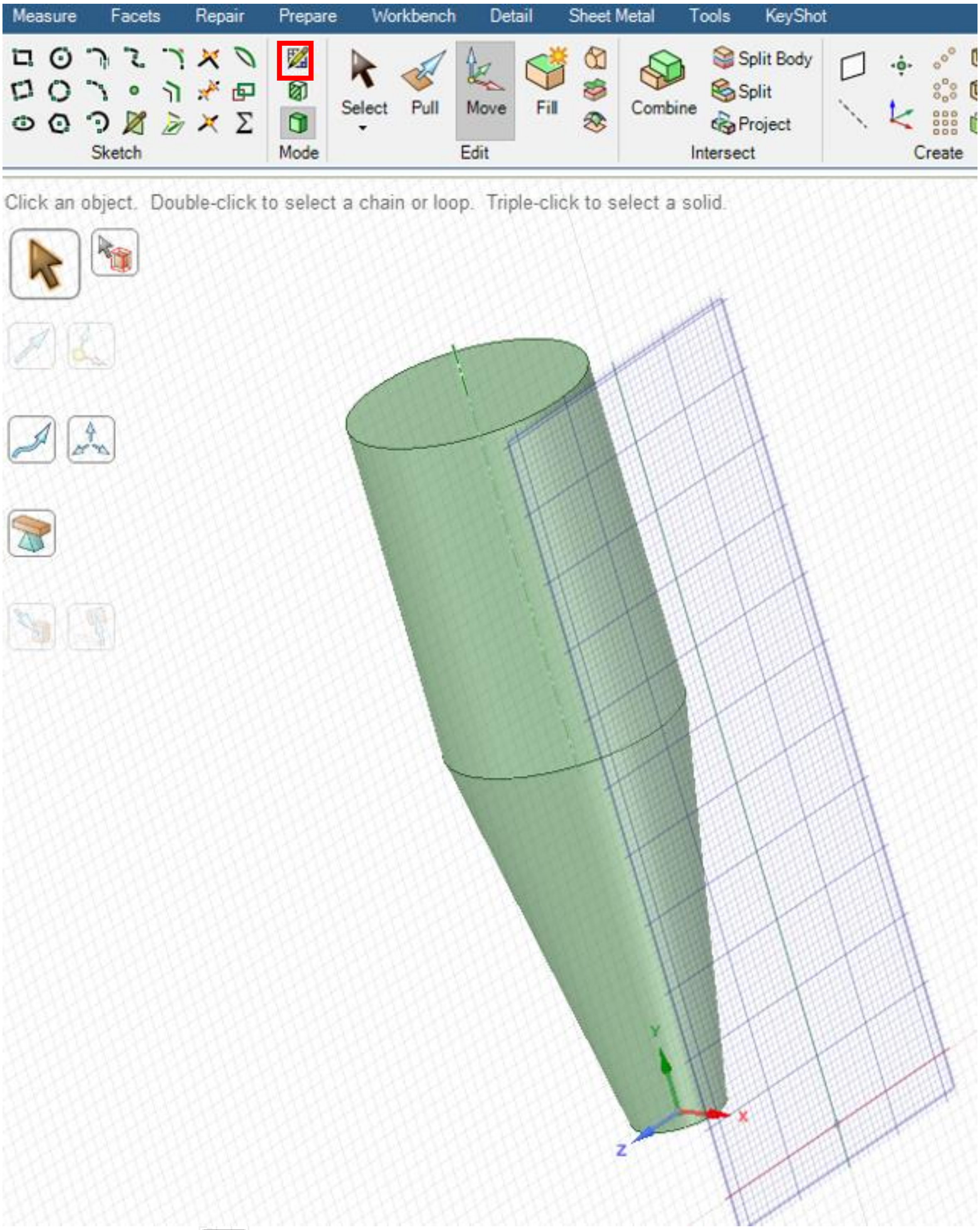
10) Select *Move* and LMB select the plane you just created

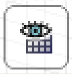


11) LMB grab the blue arrow and move the coordinate system. In the dimension edit box enter 550 mm and confirm *Enter*.

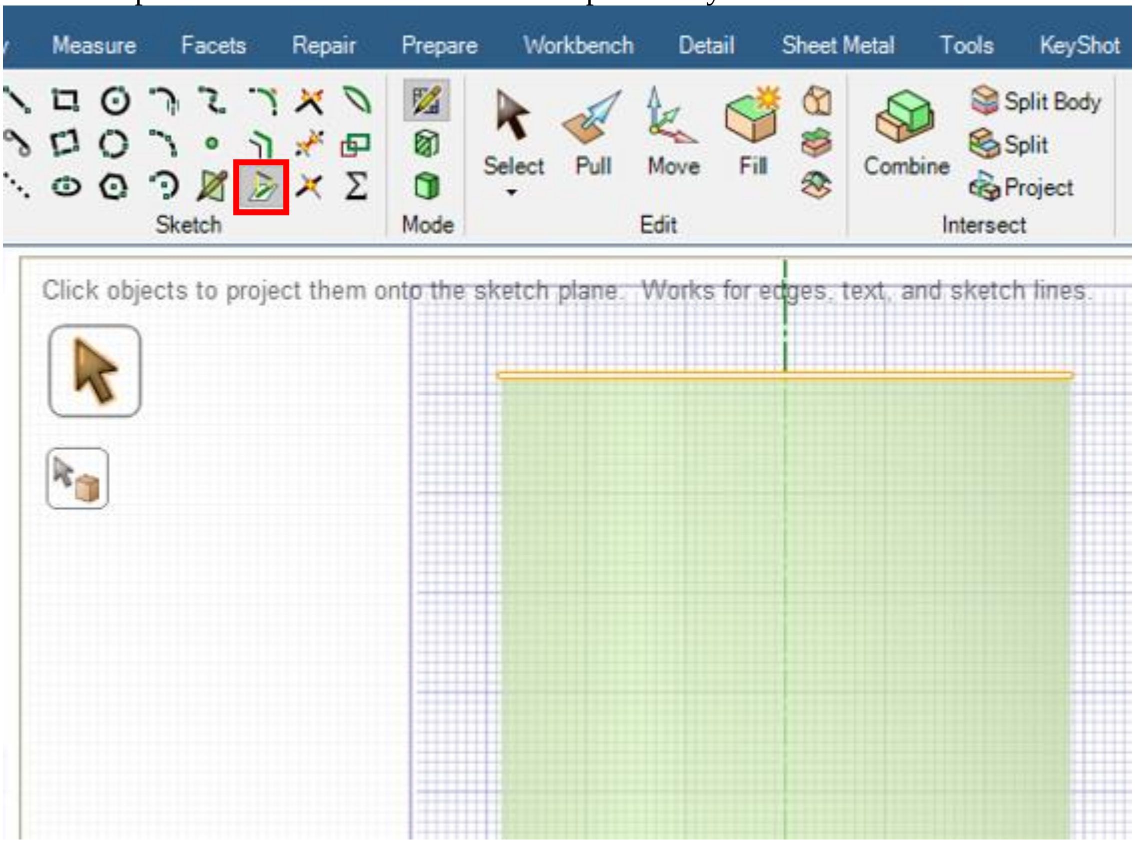


12) Select the sketch drawing icon and LMB select the plane.

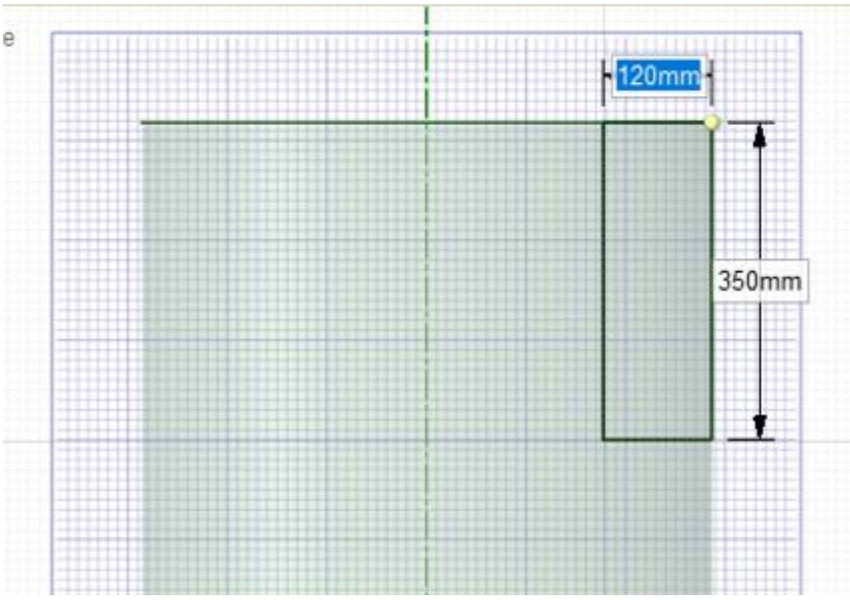


13) Click *Plan View*  to rotate the drawing plane parallel to the screen (you can also do this by pressing *Shift* + *v*).

14) To get the reference point, select the projection icon on the sketch  and LMB point to the horizontal line at the top of the cyclone

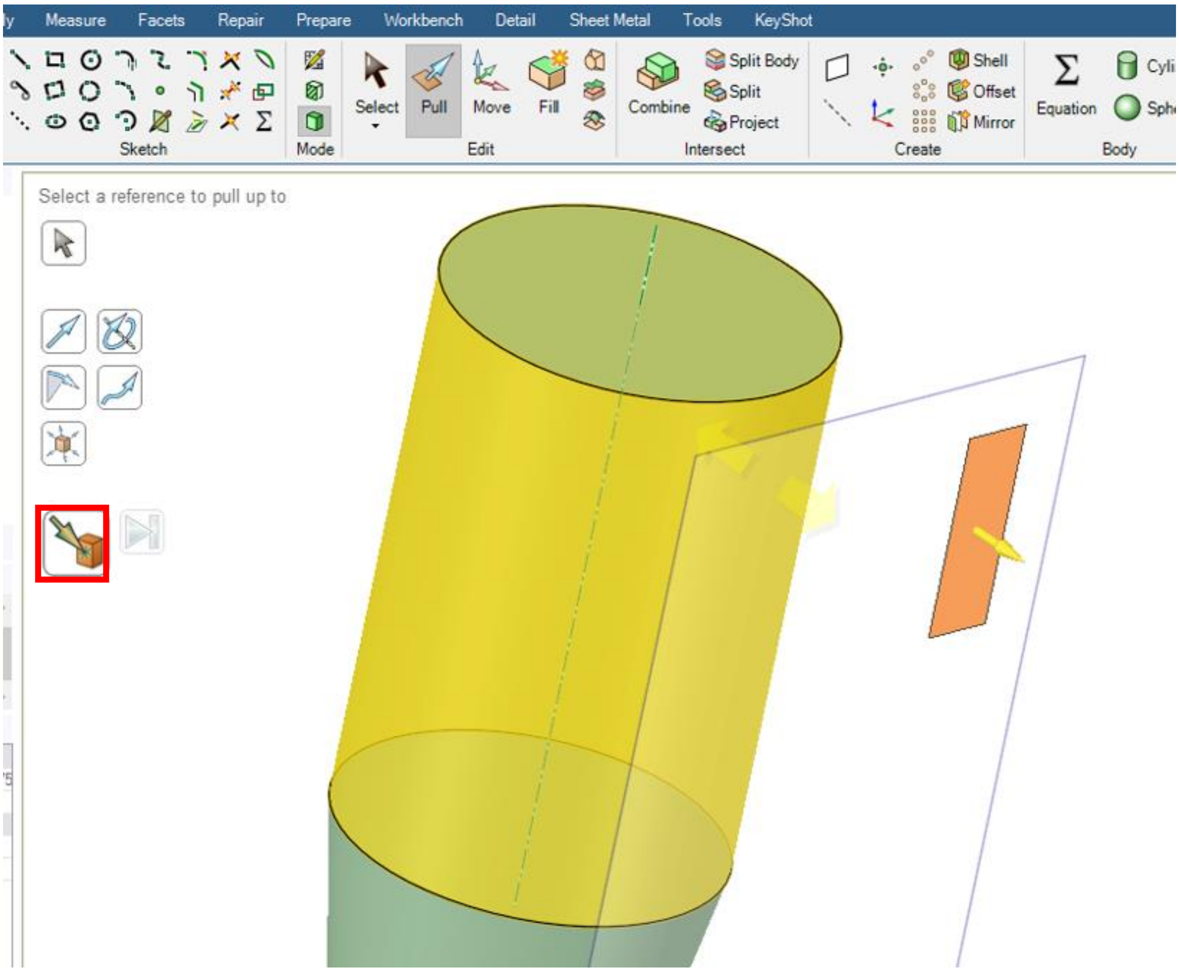


15) Draw a rectangle measuring 120 x 350 mm, the top of which coincides with the right end of the line

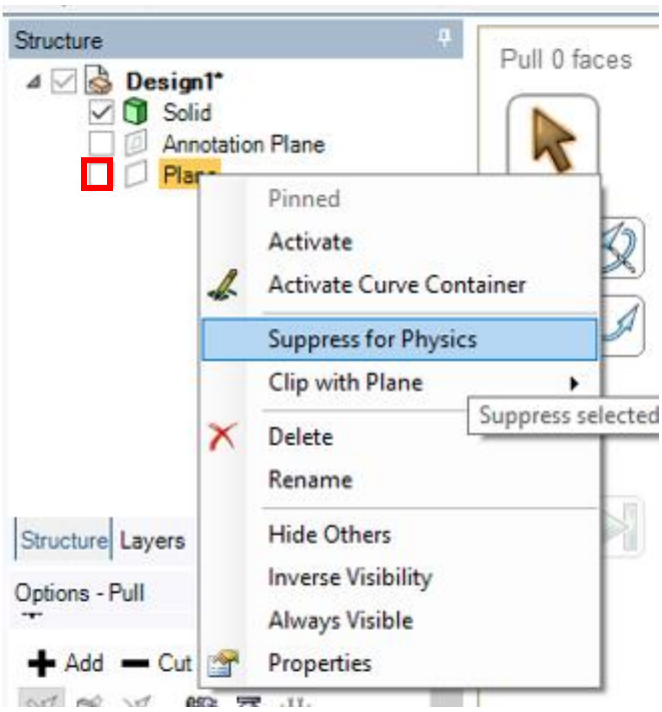


16) Point to the LMB horizontal line at the top of the cyclone and press *Delete* to delete it

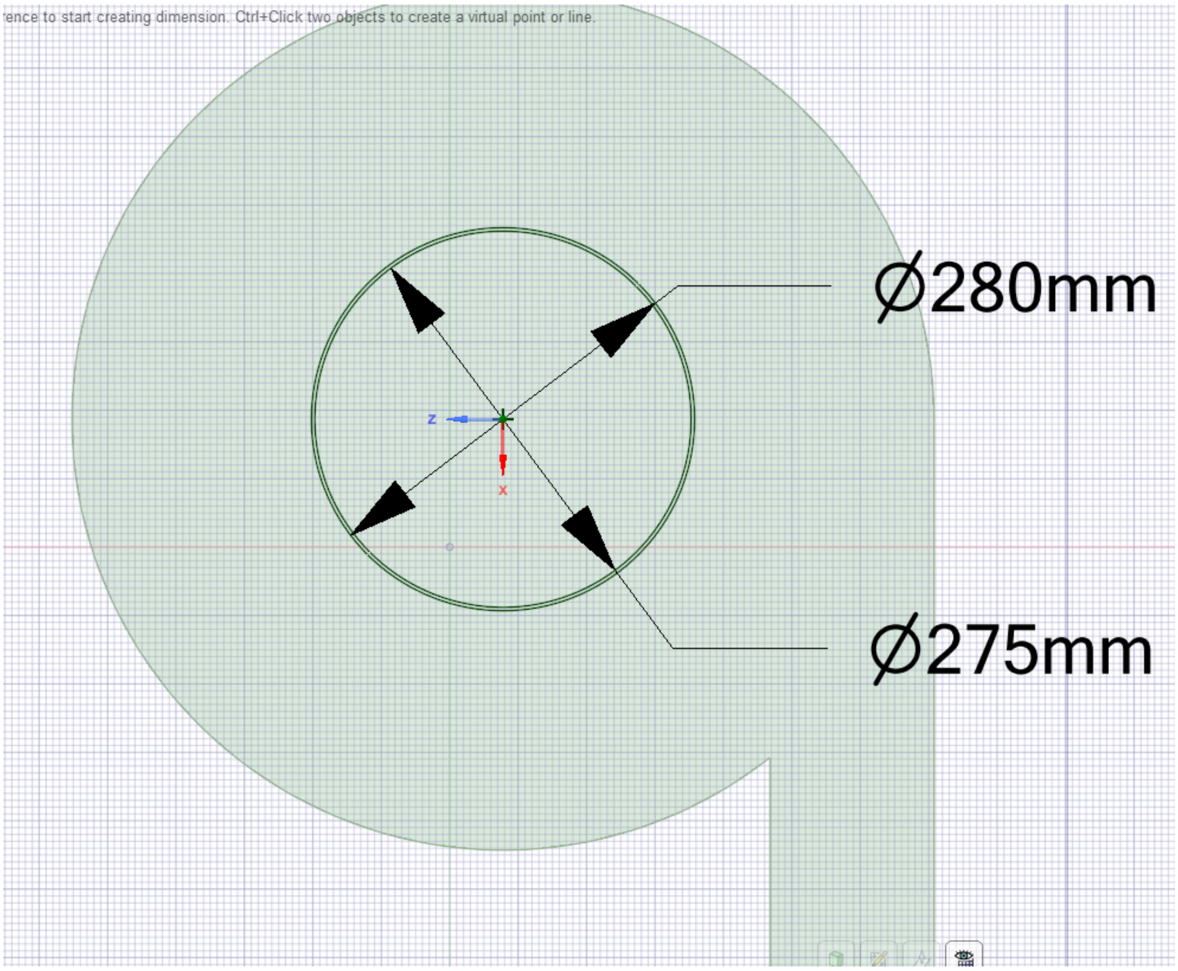
17) Return to 3D view. Using *Pull* point to the rectangle drawn and select the option *Up to* . Then select the yellow cylindrical surface marked in the drawing below.



18) Turn off plane visibility and right click (RMB) to select an option *Suppress for Physics*

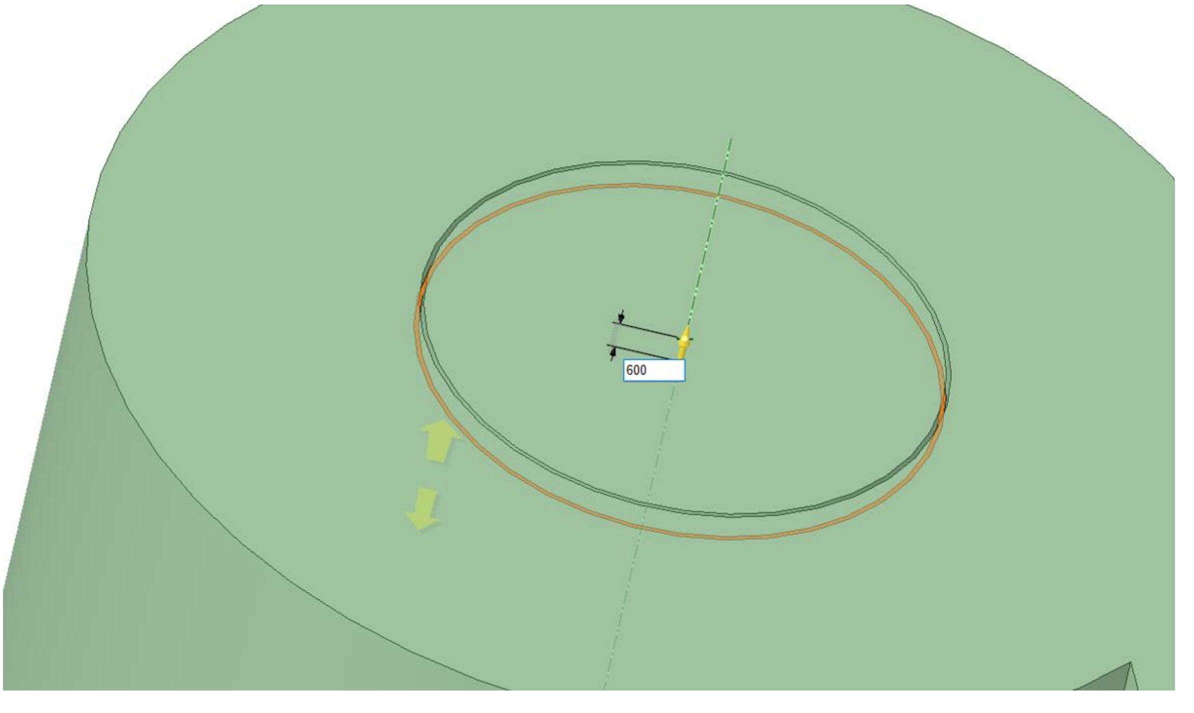


19) On the upper surface of the cyclone, draw two concentric circles with diameters 280 and 275 mm.

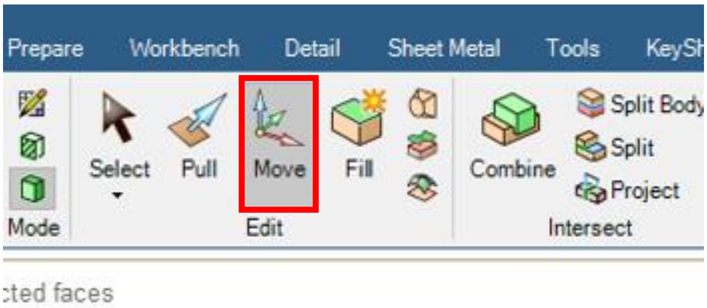


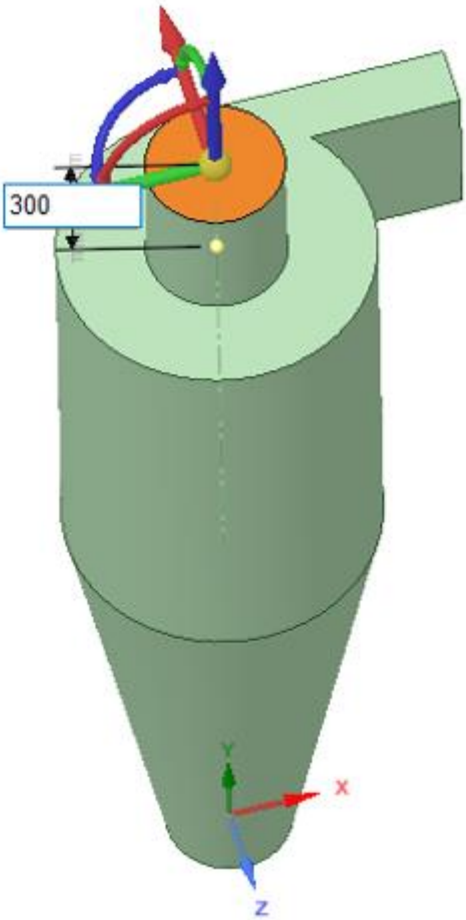
20) Using *Pull* indicate the LMB surface as shown below and select the *Cut* option from the popup menu

21) Move the LMB cursor deep into the material and select 600 mm in the dimension edit field and confirm *Enter*.

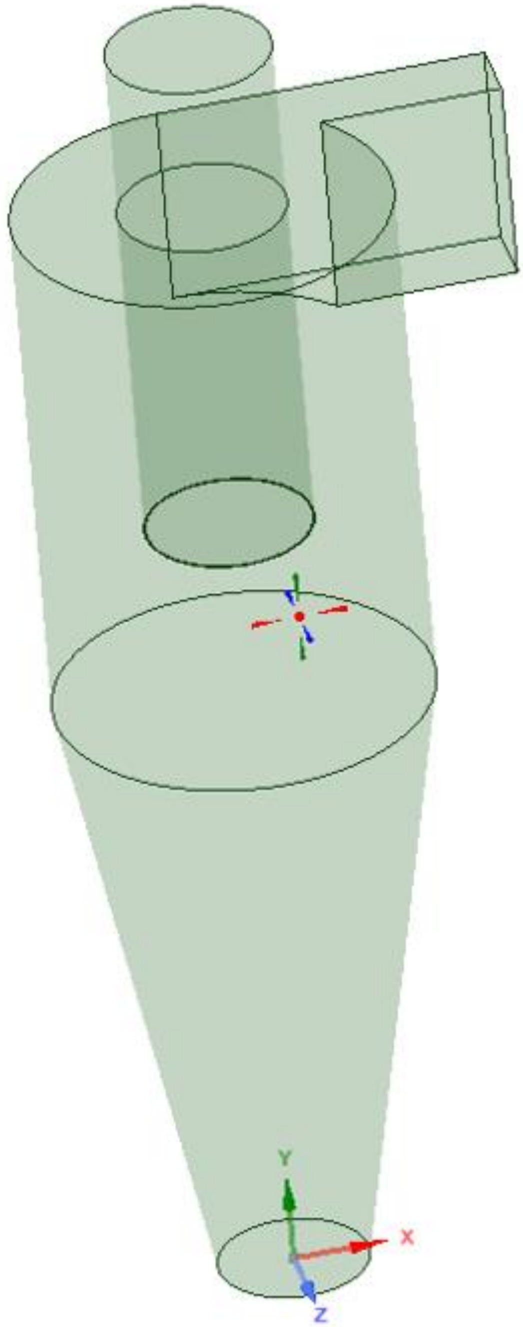


22) Using *Move* point the circle on the upper surface of the cyclone and pull it up on 300 mm





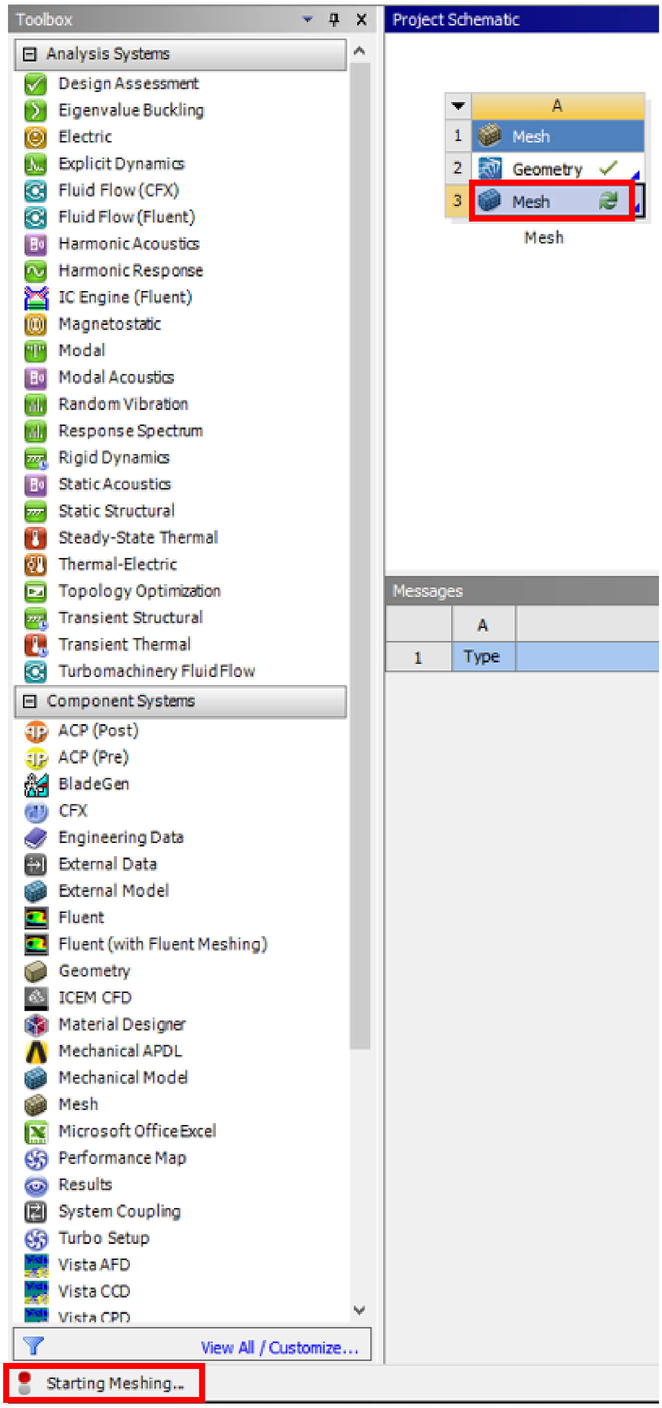
23) Cyclone geometry is ready



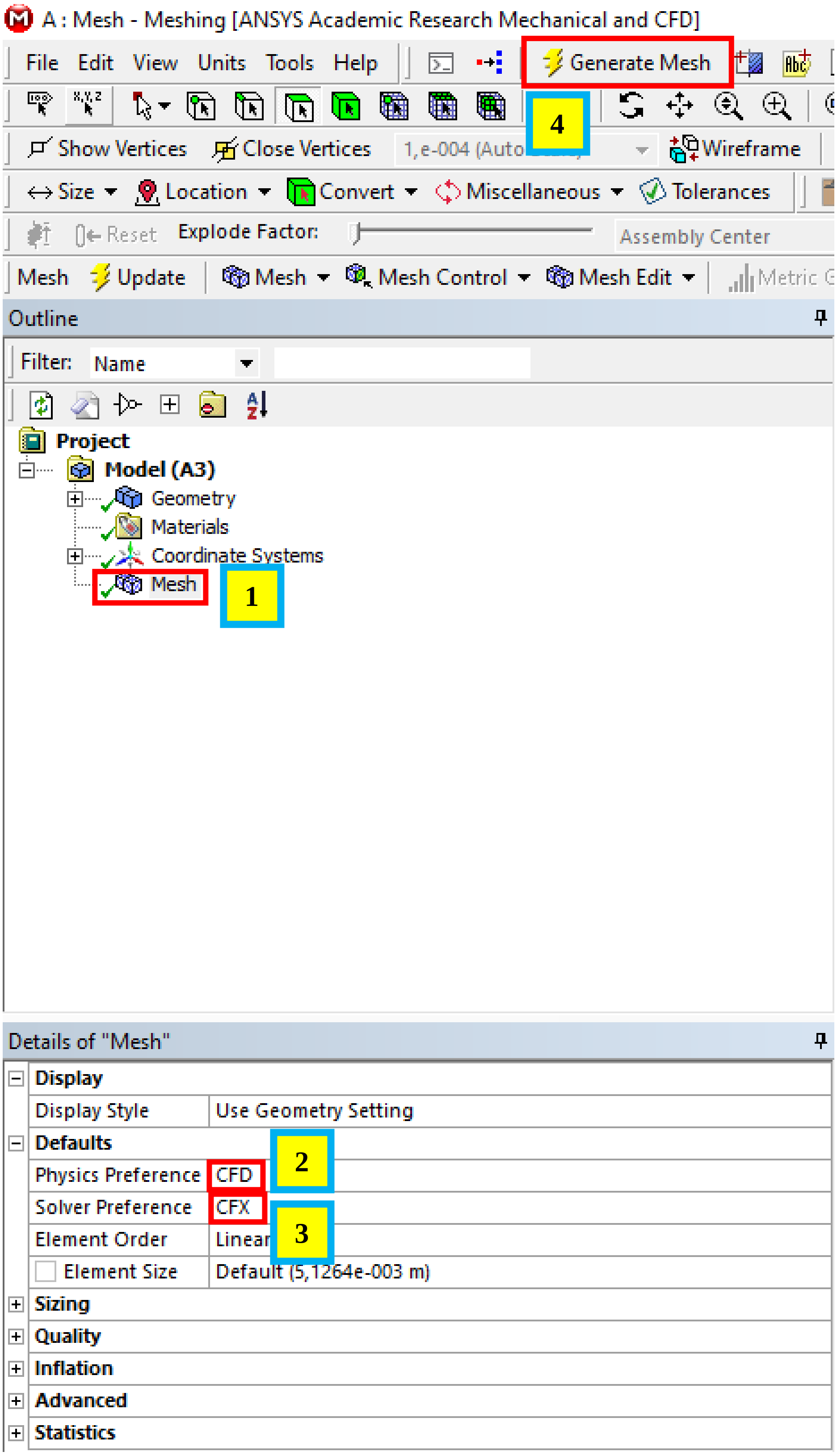
24) Close *Spaceclaim* and save project in *Workbench* using *Ctrl + s*

2.2. NUMERICAL MESH

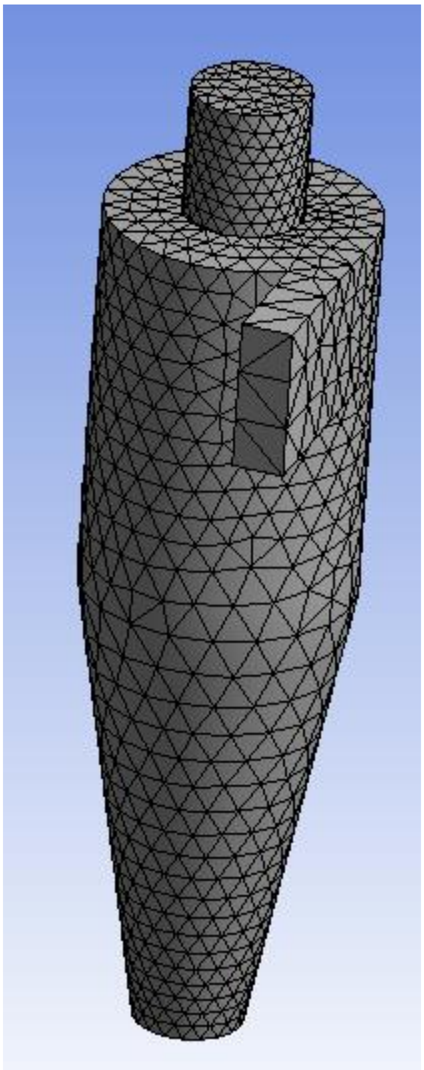
1) Open *Ansys Meshing* by double-click LMB on *Mesh*



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



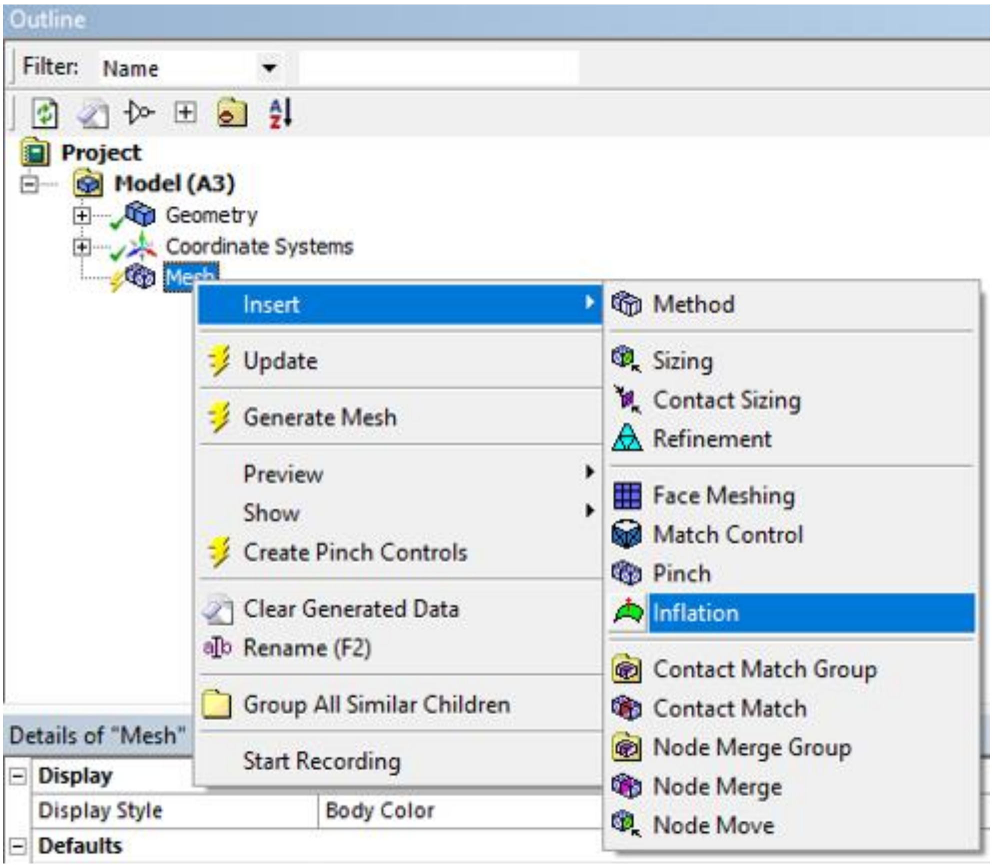
The default mesh is not valid. The mesh should be edited.



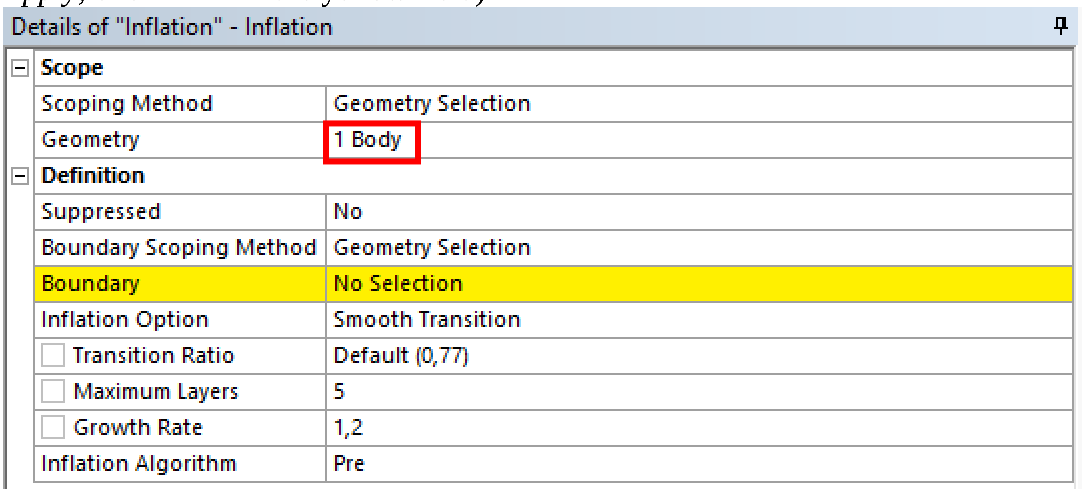
3) In *Details of „Mesh”* -> *Sizing* change value *Max Face Size* into 0,03 m

Details of "Mesh"	
Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	CFX
Element Order	Linear
Sizing	
Size Function	Curvature
Max Face Size	0,03
Mesh Defeaturing	Yes
Defeature Size	Default (1,5e-004 m)
Growth Rate	Default (1,20)
Min Size	Default (3,e-004 m)
Max Tet Size	Default (6,e-002 m)
Curvature Normal Angle	Default (18,0 °)
Bounding Box Diagonal	2,26830 m

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

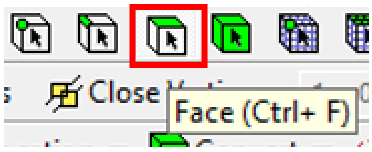


LMB select the cyclone and click *Scope* -> *Geometry* -> *Apply* (if you don't see *Apply*, click LMB in the yellow field)

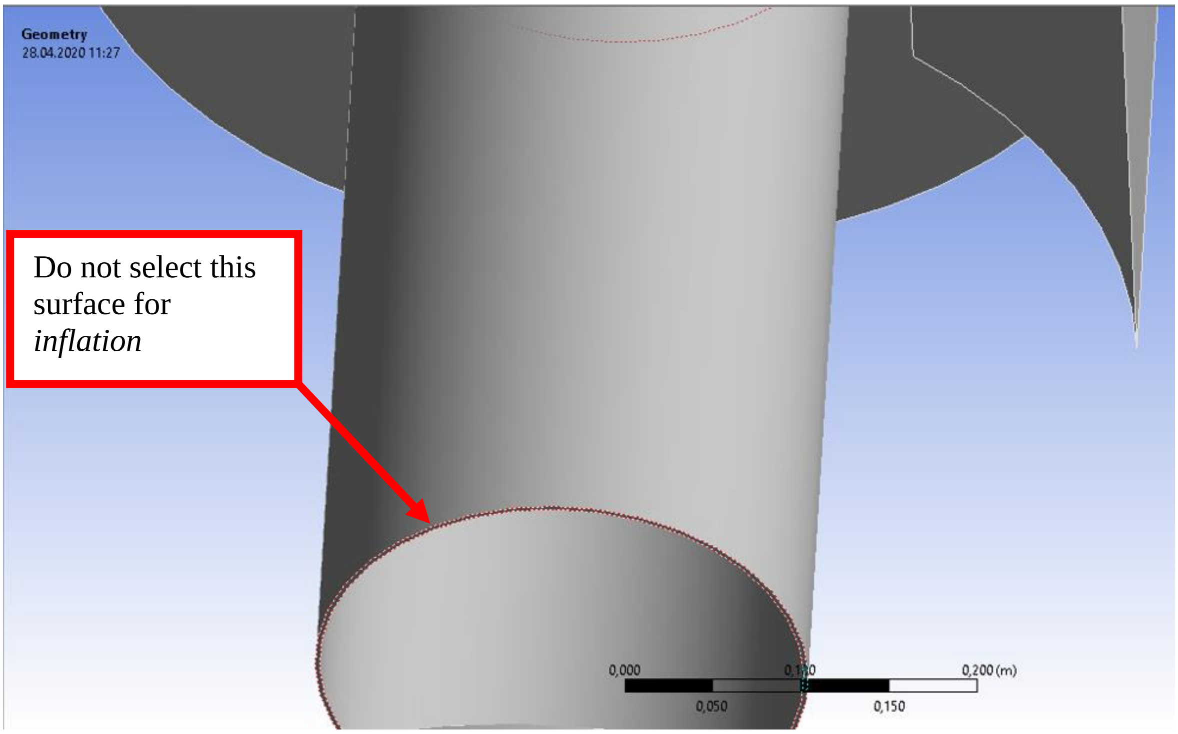

Details of "Inflation" - Inflation

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	No Selection
Inflation Option	Smooth Transition
Transition Ratio	Default (0,77)
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre

5) Then change the filter to face selection



Indicate all the walls of the cyclone except the small flat inner surface indicated in the figure below



In *Definition* in the *Boundary* field confirm *Apply*

Details of "Inflation" - Inflation

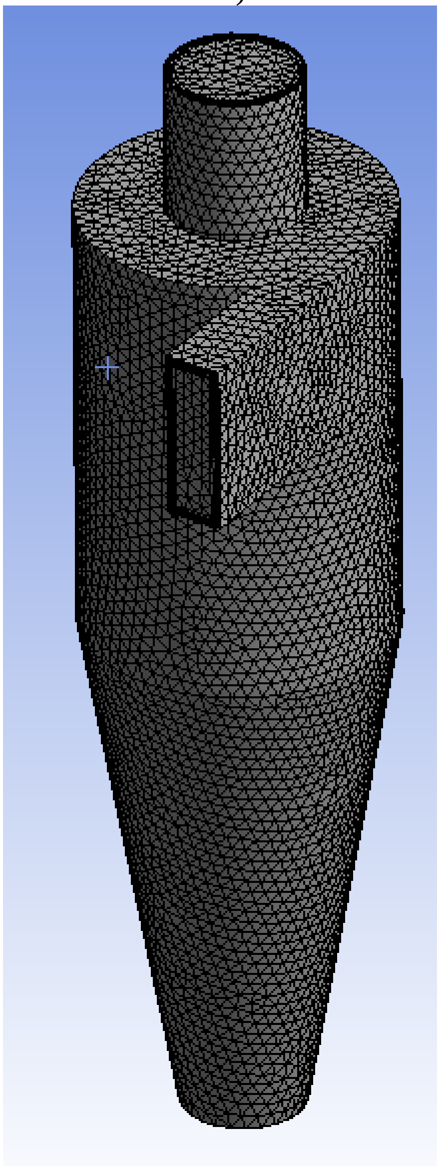
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	Apply	Cancel
Inflation Option	Smooth Transition
Transition Ratio	Default (0,77)
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre

Alter *Transition Ratio* into 0,2

Details of "Inflation" - Inflation

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	8 Faces
Inflation Option	Smooth Transition
Transition Ratio	0,2
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre

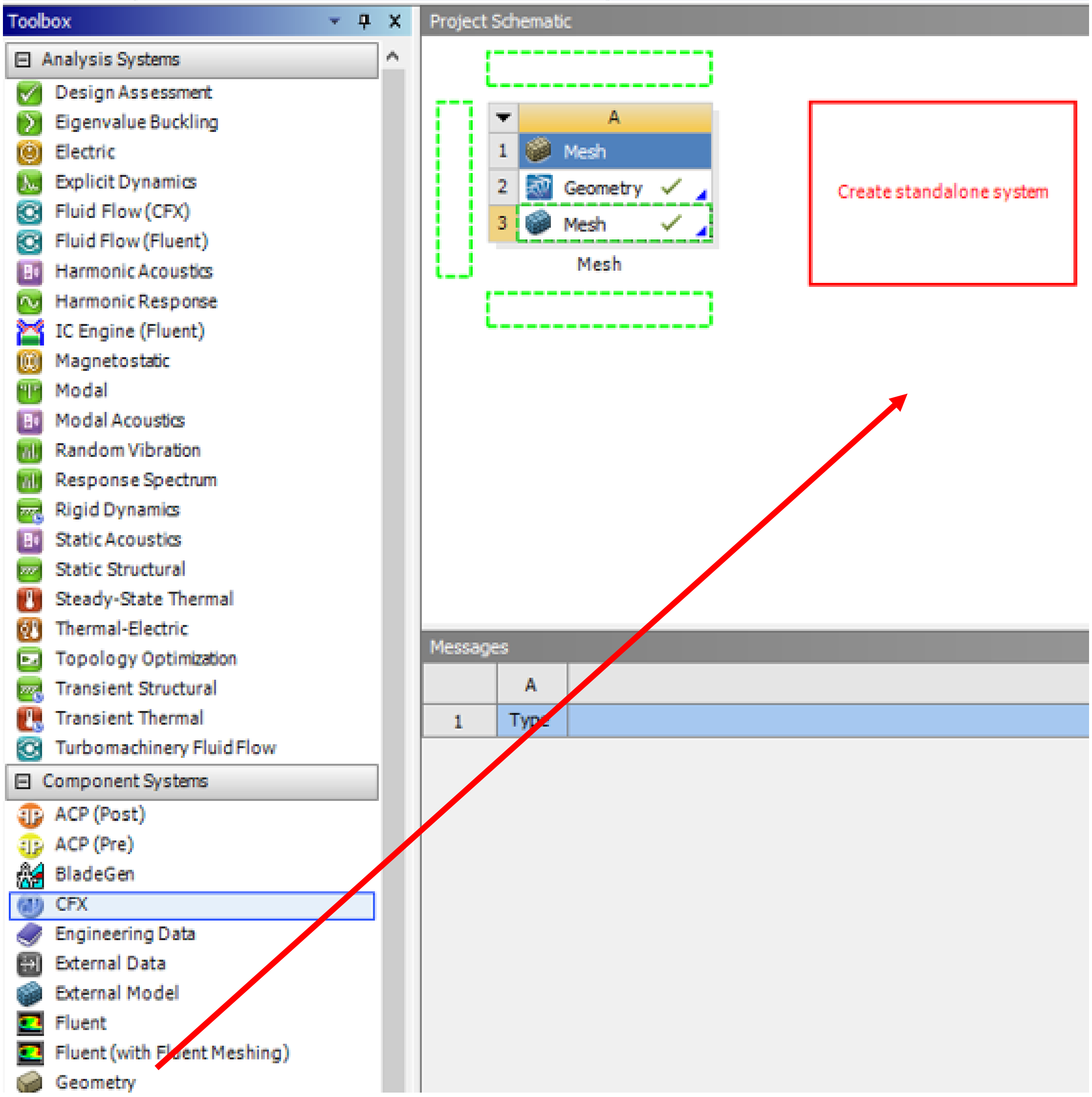
Click *Generate Mesh* and check the generated mesh (if the grid is not visible, click LMB on *Mesh* in the tree on the left).



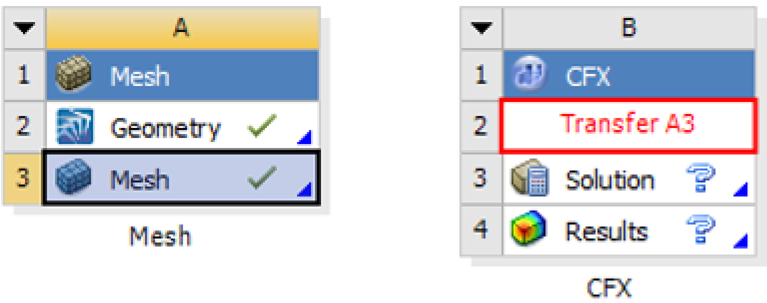
6) The last step is to name the volumes and surfaces. Use *Create Named Selection* to give the following names (if you don't remember how to use *Create Named Selection*, check the previous instructions):
   a. Solid volume - *Fluid_domain*
   b. Inlet surface (rectangle) – *inlet*
   c. Particle outlet surface (circle at the bottom of the cyclone) - *outlet1*
   d. Clean air outlet surface (circle at the top of the cyclone) - *outlet2*
   Wall surfaces are not important here, i.e. *no-slip Wall* condition will be applied to them. This is the default condition in Ansys CFX, so you do not need to name all faces to later set the boundary condition "manually". If it is necessary to give a non-default condition (e.g. heat flux) it would be good in *Ansys Meshing* to name all wall surfaces with one name, e.g. *walls*.
7) Close *Ansys Meshing* and save project in *Workbench.*

2.3. NUMERICAL MODEL

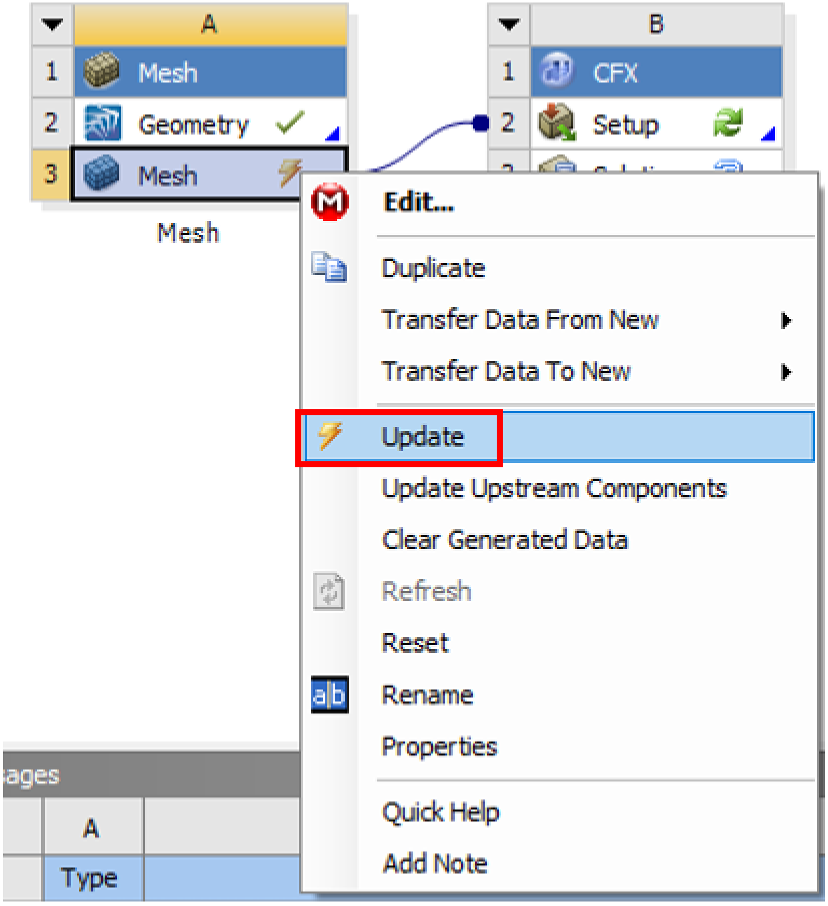
1) Insert *CFX*



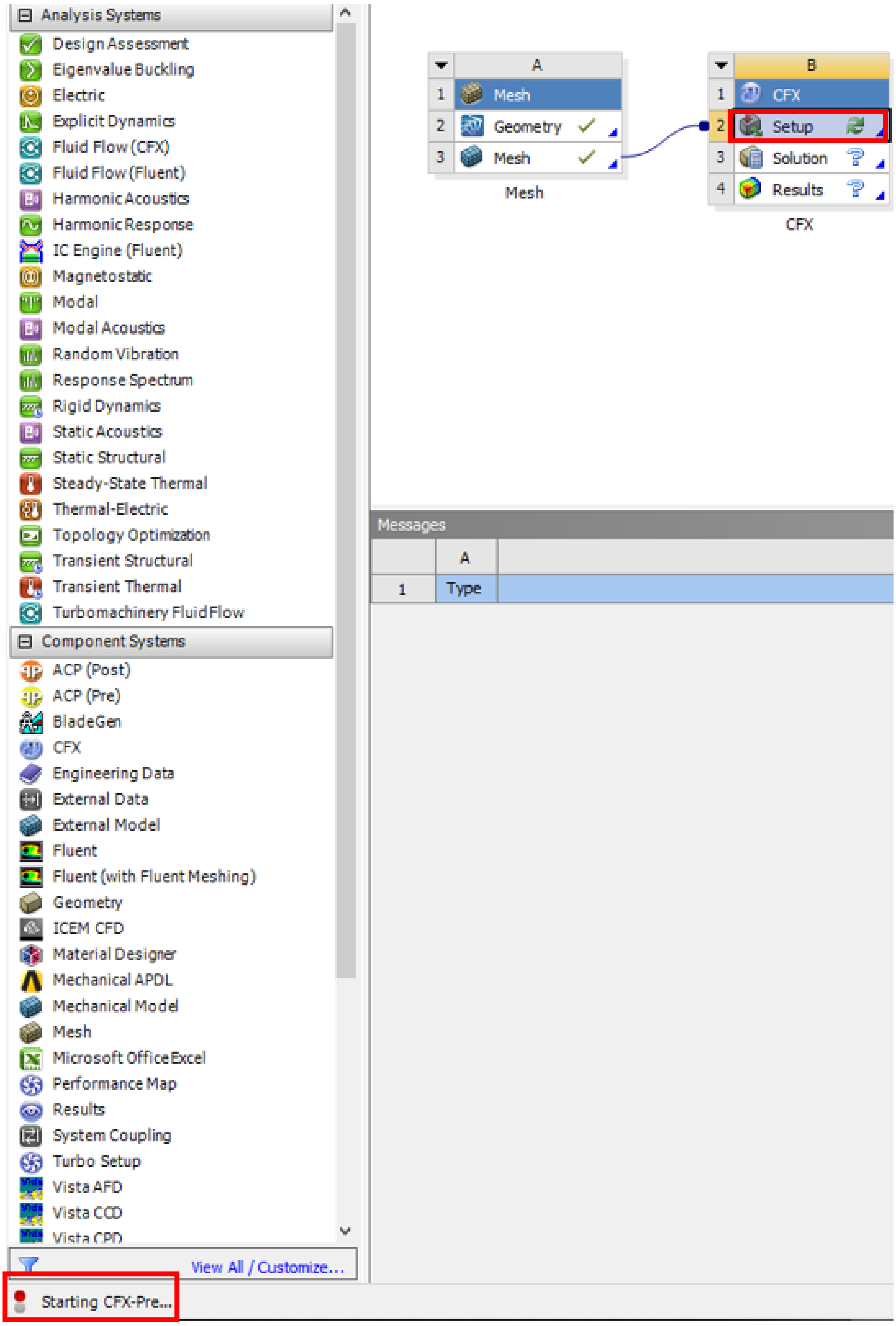
To connect the *Mesh* module with *CFX*, grab LMB *Mesh* (below) and drag it to *Setup* until the *Transfer A3* box appears, then release LMB - the connection has been created



Click RMB on *Mesh* and select *Update*



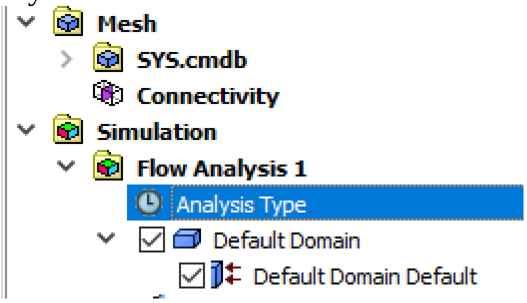
Double-click *Setup* to run *Ansys CFX*



2) To calculate the effect of particles on a continuous fluid, typically 100 to 1000 particles are required. However, if accurate information about the volume concentration of particles or local forces on the walls is required, then a much larger number of particles should be modeled. When creating a domain, either *Full* (Coupled) or *One-way Coupling* can be selected between the particle and the continuous phase. *Full coupling* is needed to predict the effect of particles on a continuous phase flow field, but has greater computational requirements

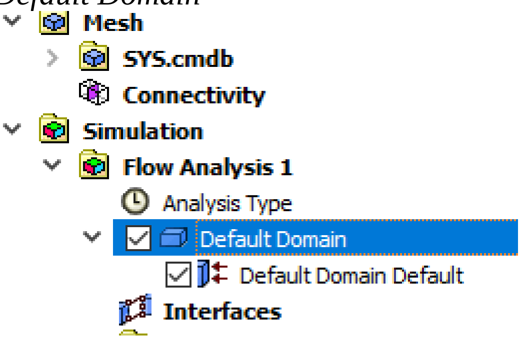
than a one-way connection; *One-way coupling* simply predicts the particle paths based on the flow field, but without affecting the flow field [1].

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

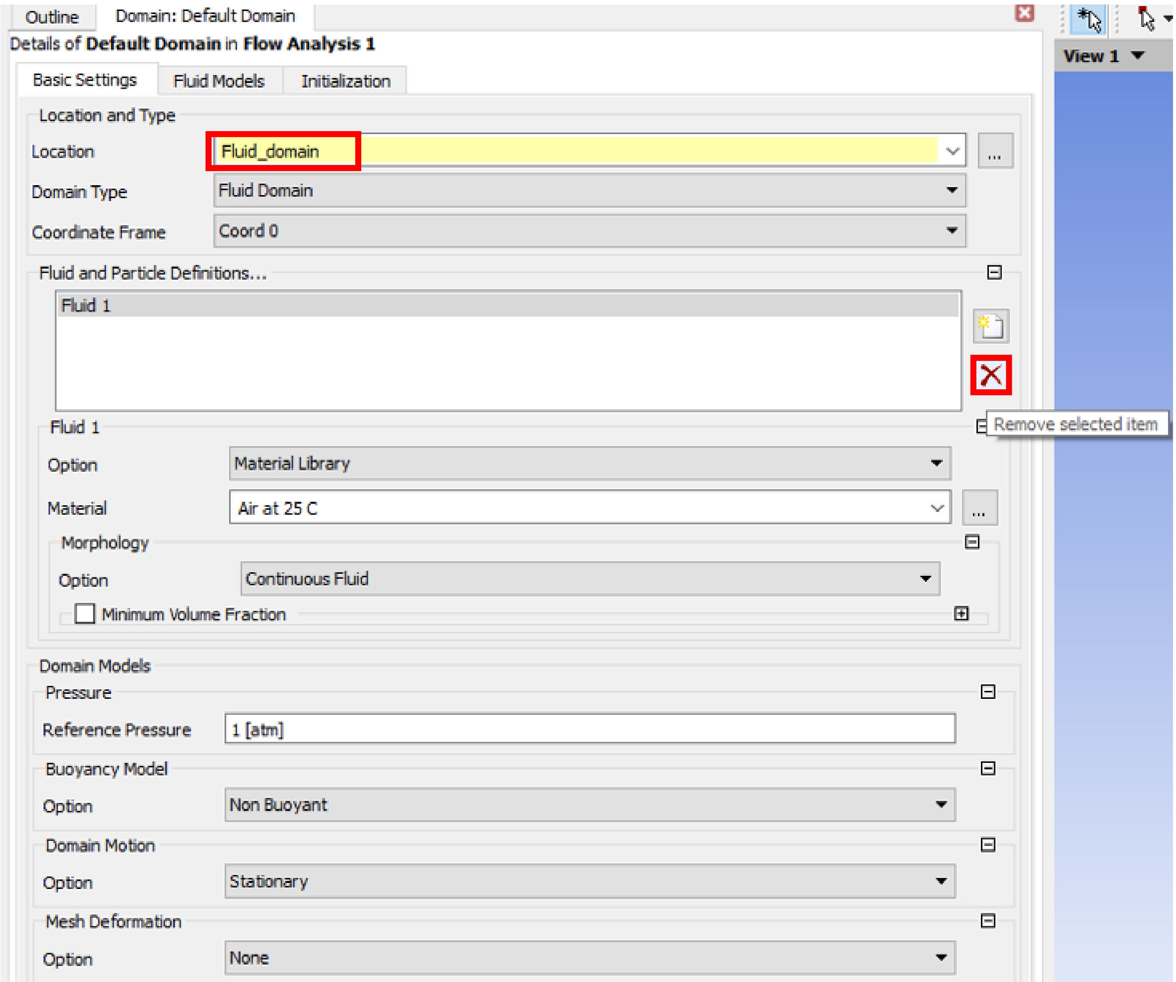


Apply the following settings and confirm *OK*.

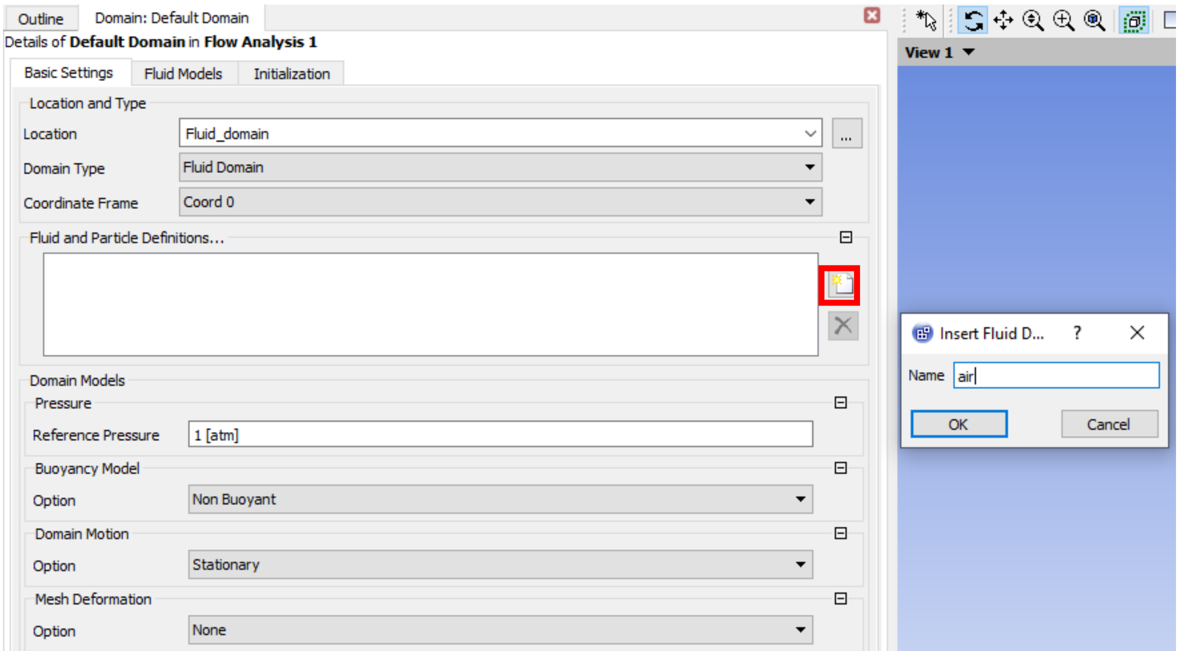
4) Double-click LMB *Default Domain*



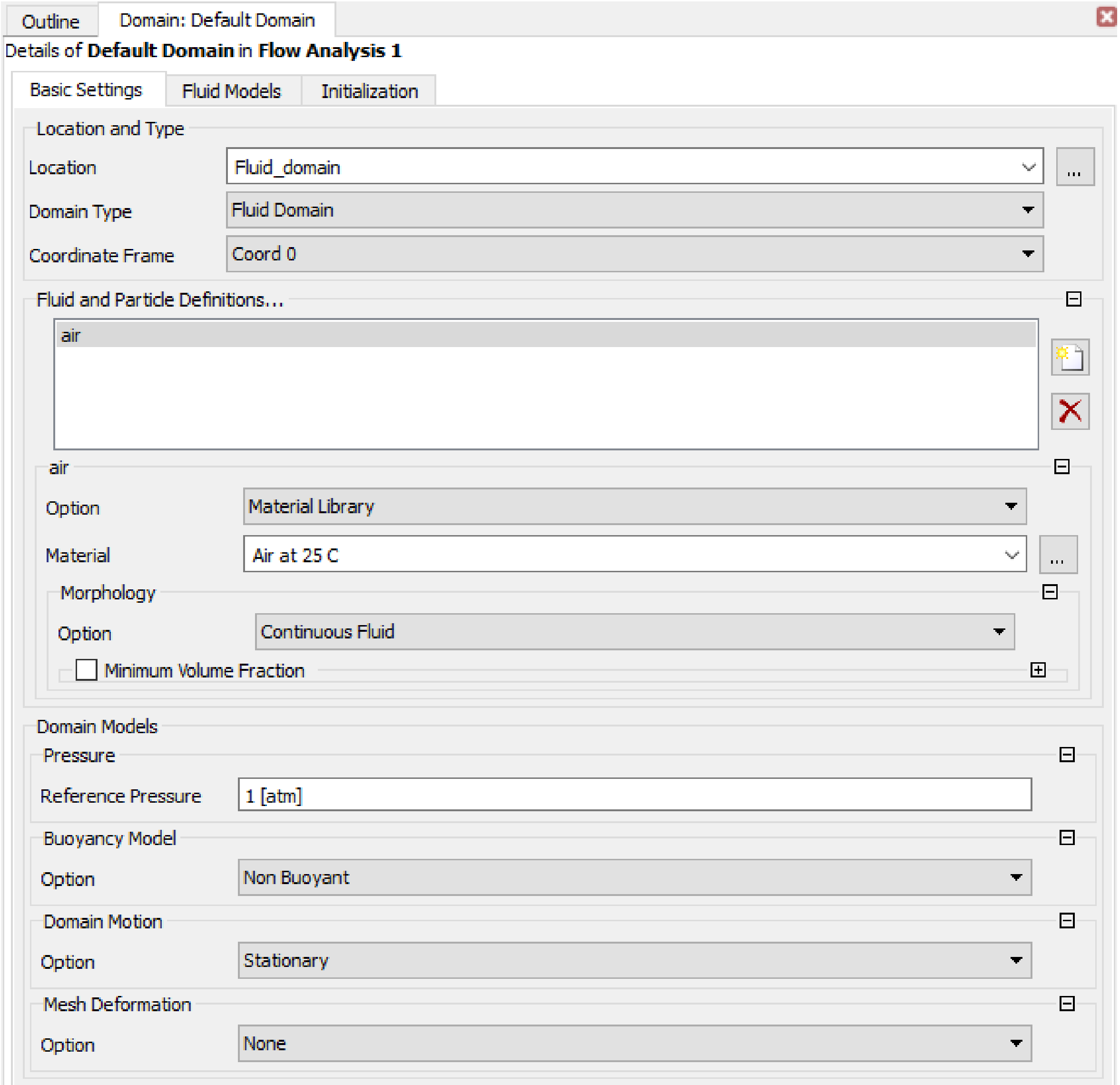
Apply the following settings and delete the default material



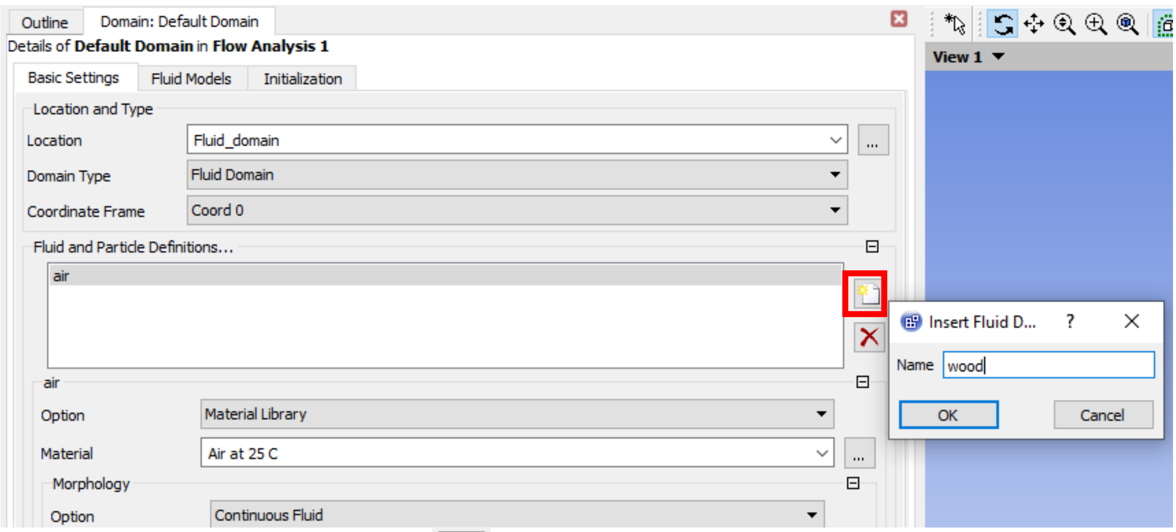
5) Add the first material and name it *air*



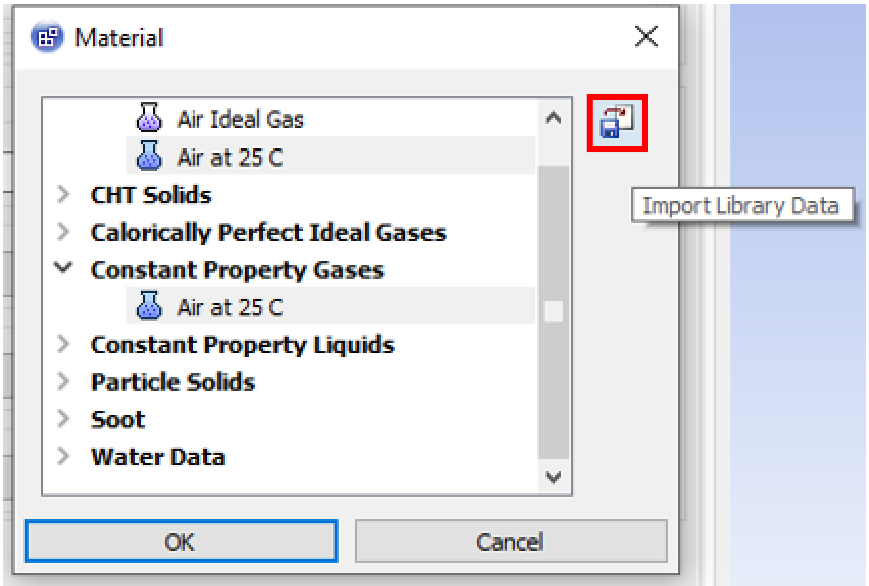
6) As a material select *Air at 25 C*



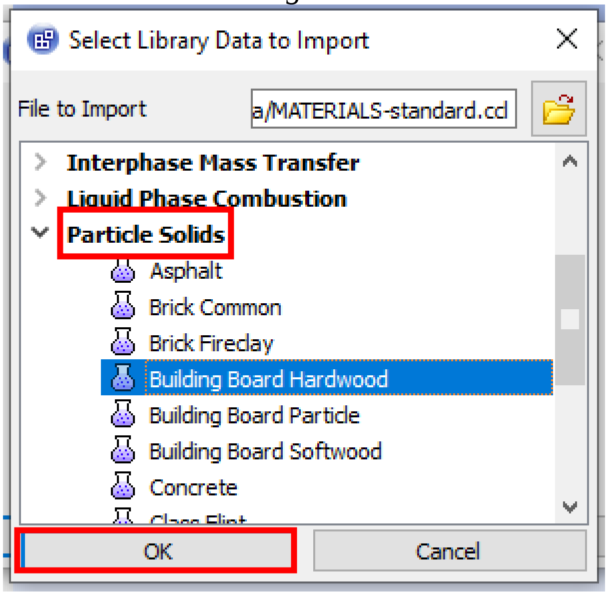
7) Add a second material and name it *wood*



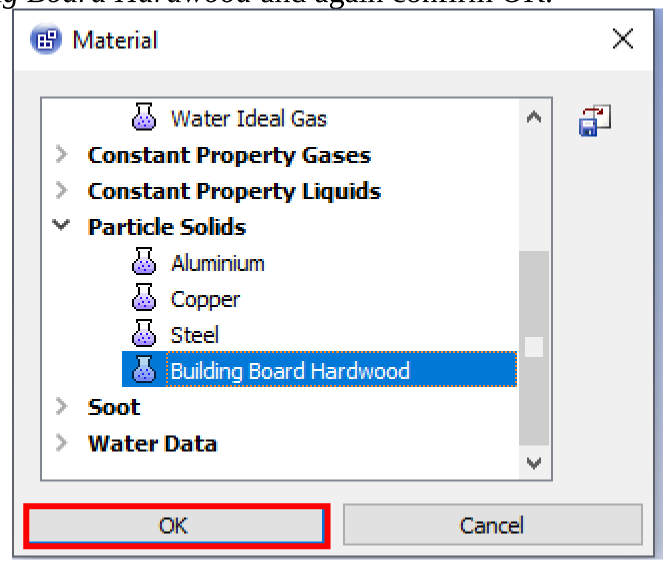
8) Select the material selection icon ..., and next *Import library data*



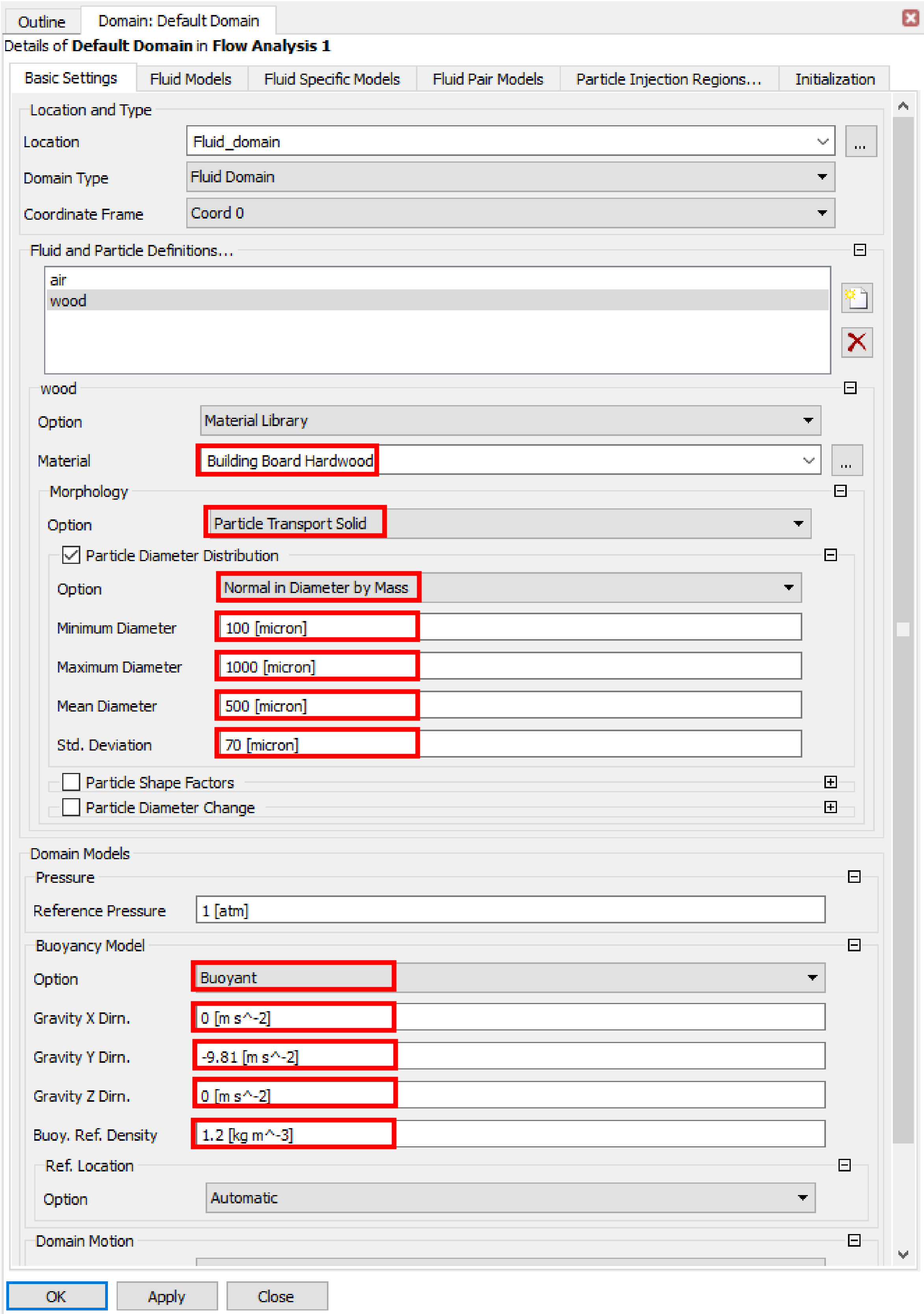
9) From tab *Particle Solids* select *Building Board Hardwood* and confirm *OK*.



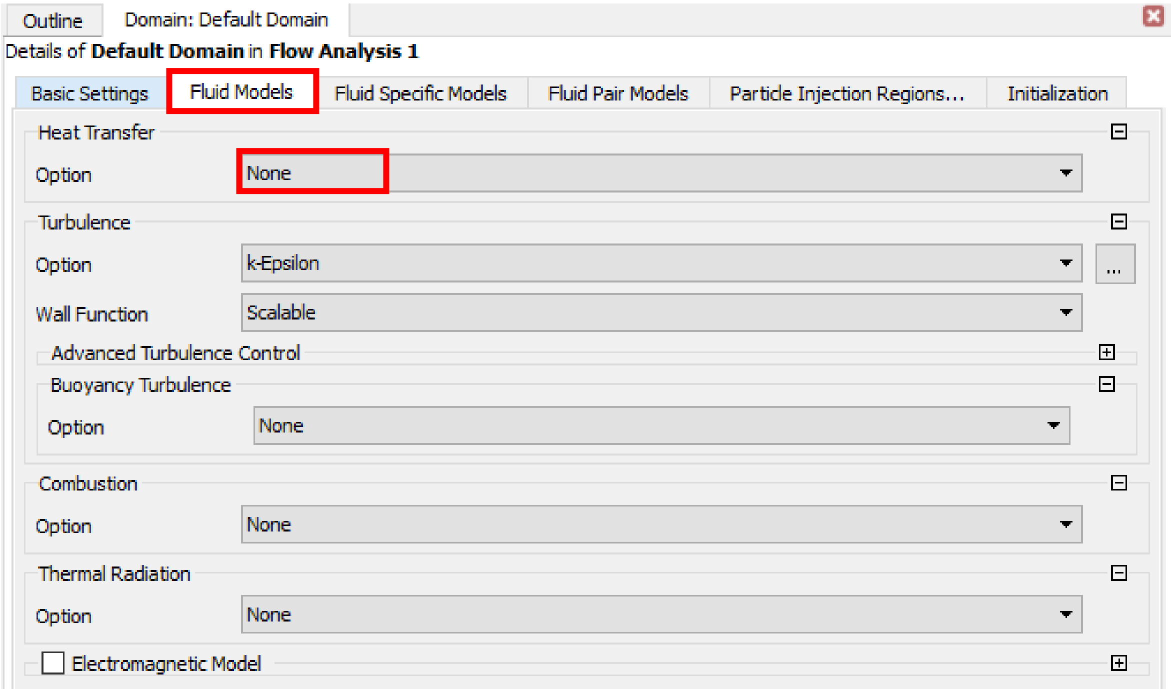
Select *Building Board Hardwood* and again confirm *OK*.



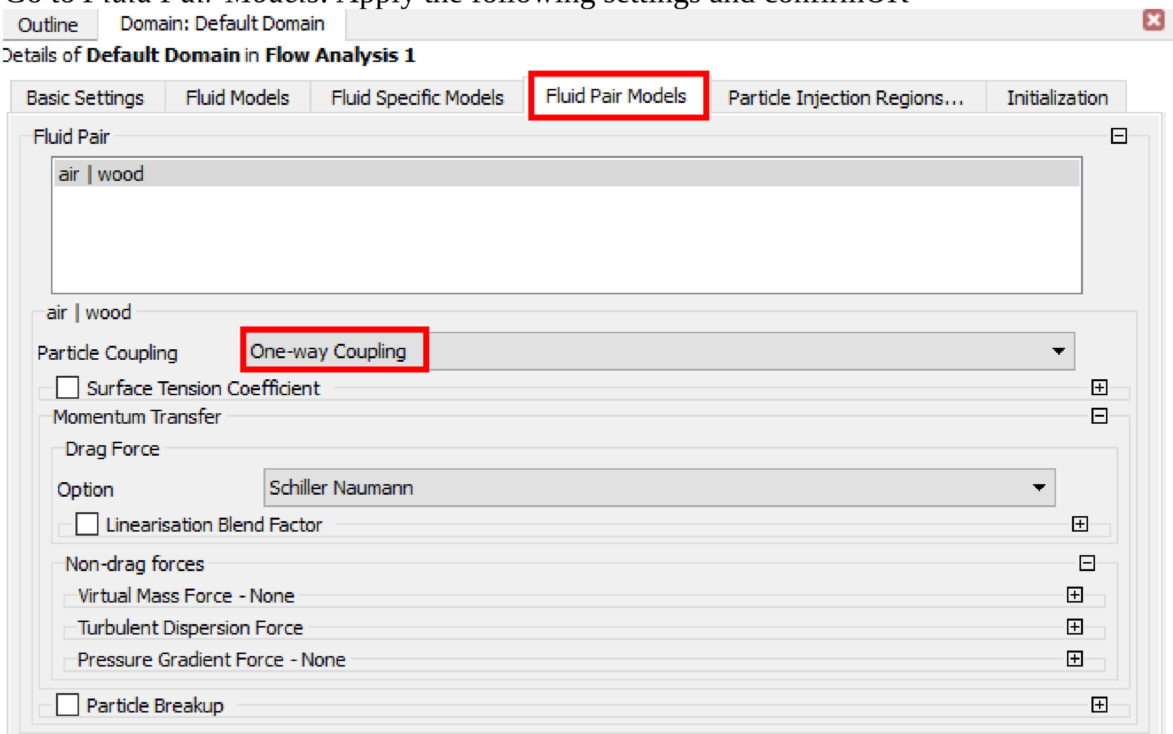
10) For *wood* apply the following settings



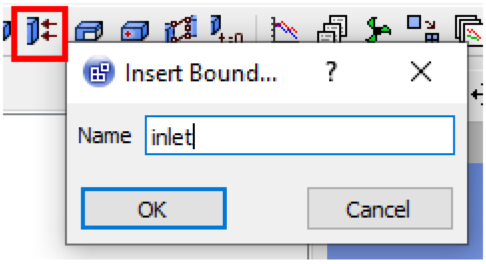
11) Go to *Fluid Models*



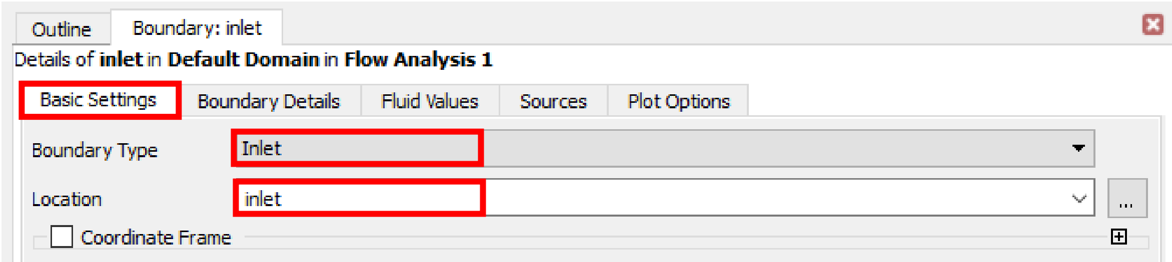
12) Go to *Fluid Pair Models*. Apply the following settings and confirm*OK*

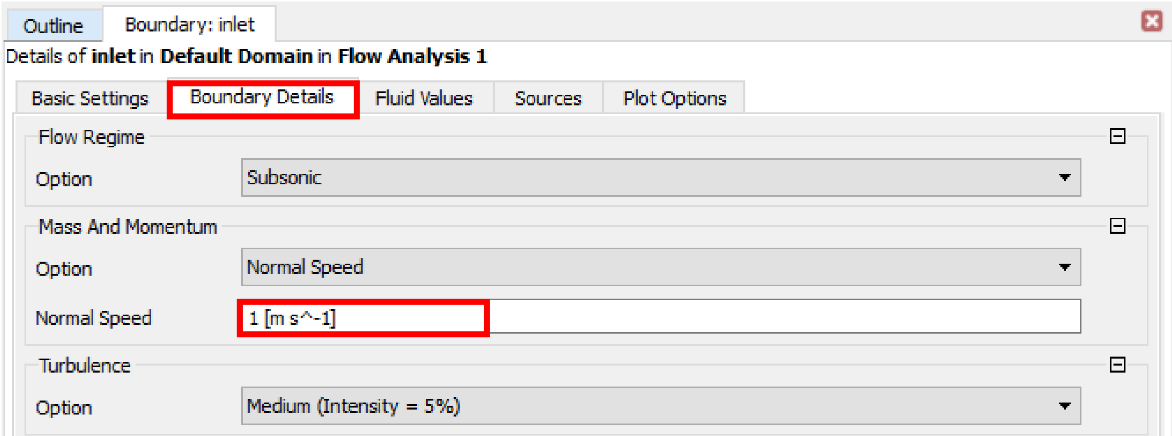


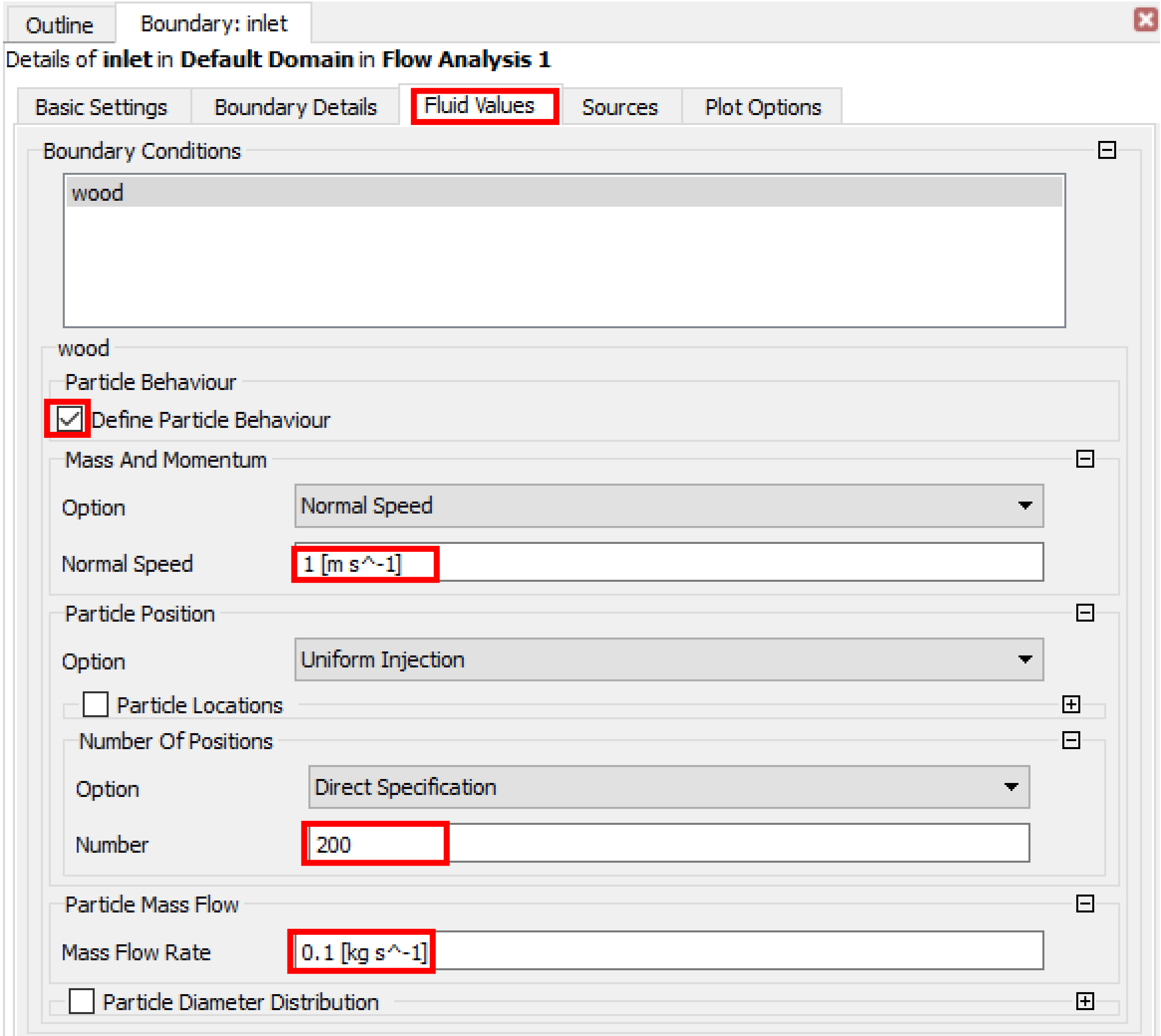
13) Create an boundary condition named *inlet*



14) Apply the following settings

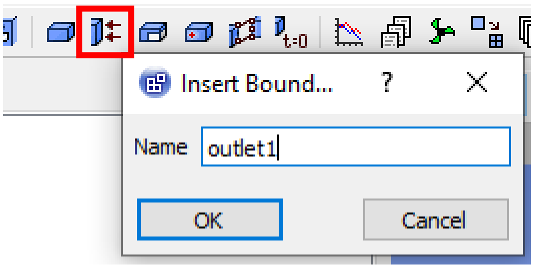

Outline
Boundary: inlet
Details of **inlet** in **Default Domain** in **Flow Analysis 1**
Basic Settings
Boundary Details
Fluid Values
Sources
Plot Options
Boundary Type
Inlet
Location
inlet
Coordinate Frame


Outline
Boundary: inlet
Details of **inlet** in **Default Domain** in **Flow Analysis 1**
Basic Settings
Boundary Details
Fluid Values
Sources
Plot Options
Flow Regime
Option
Subsonic
Mass And Momentum
Option
Normal Speed
Normal Speed
1 [m s^-1]
Turbulence
Option
Medium (Intensity = 5%)

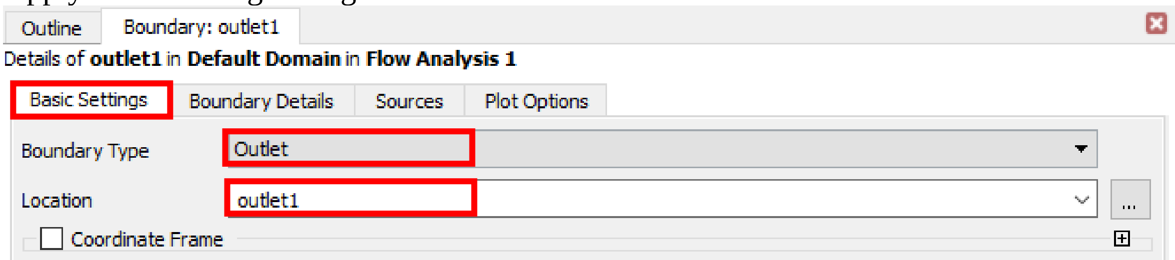


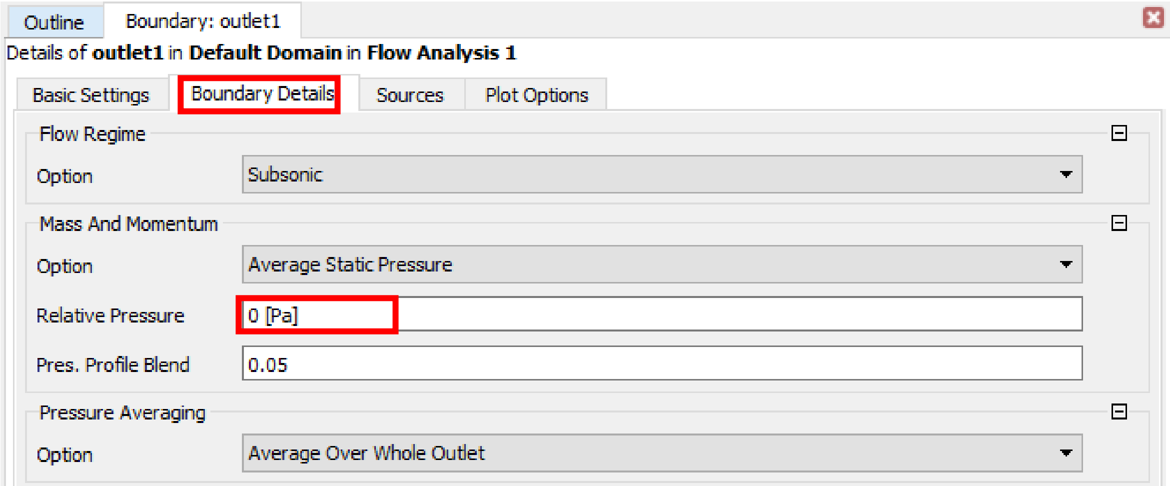
Confirm *OK*.

15) Create an boundary condition *outlet1*



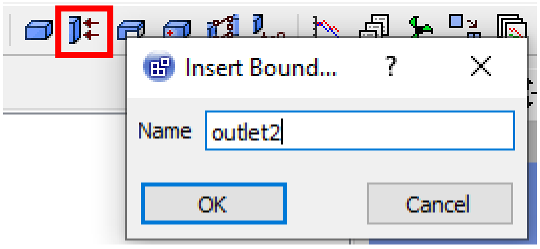
16) Apply the following settings



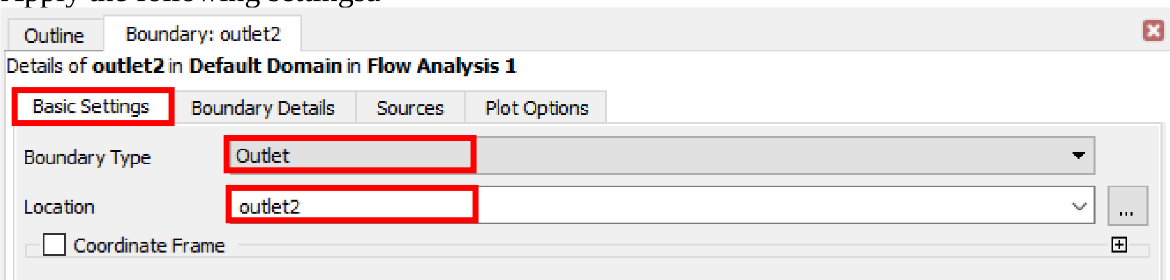


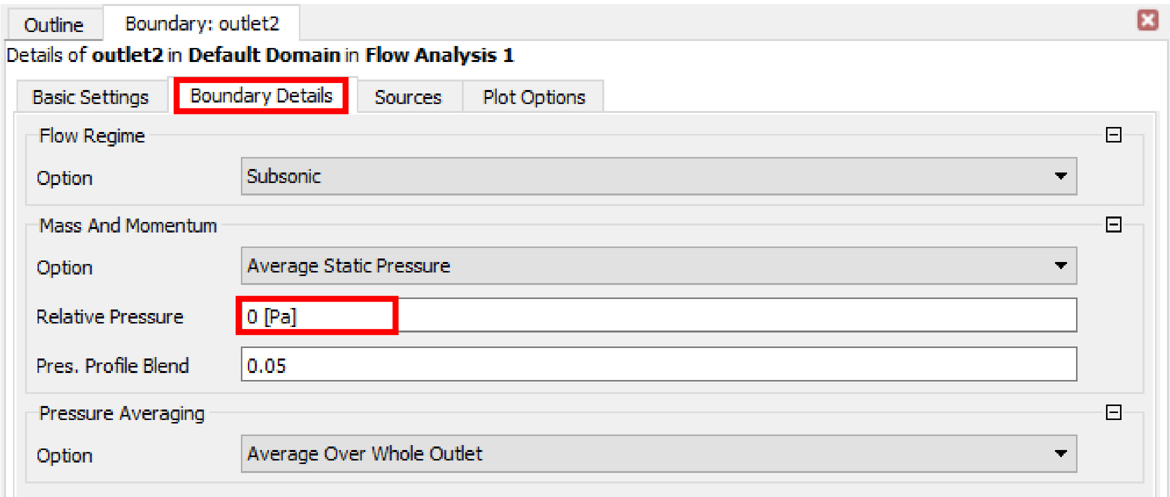
Confirm *OK*.

17) Create an boundary condition *outlet2*



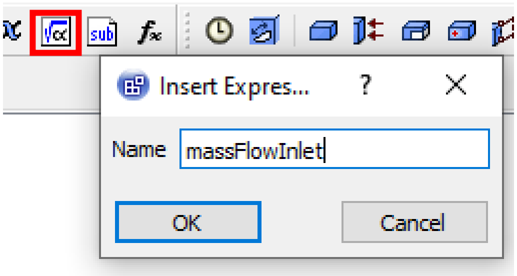
18) Apply the following settingsa



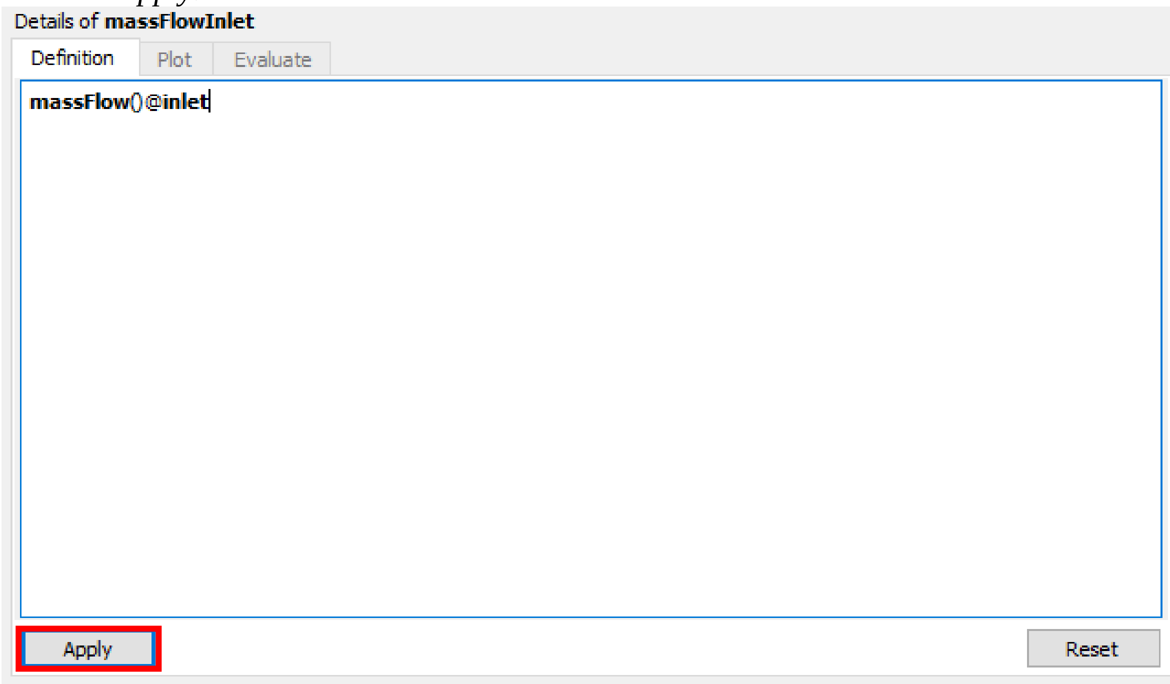


Confirm *OK*.

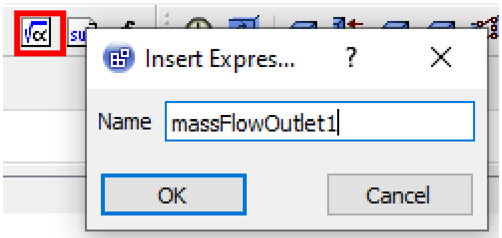
19) Create *expression* named *massFlowInlet*



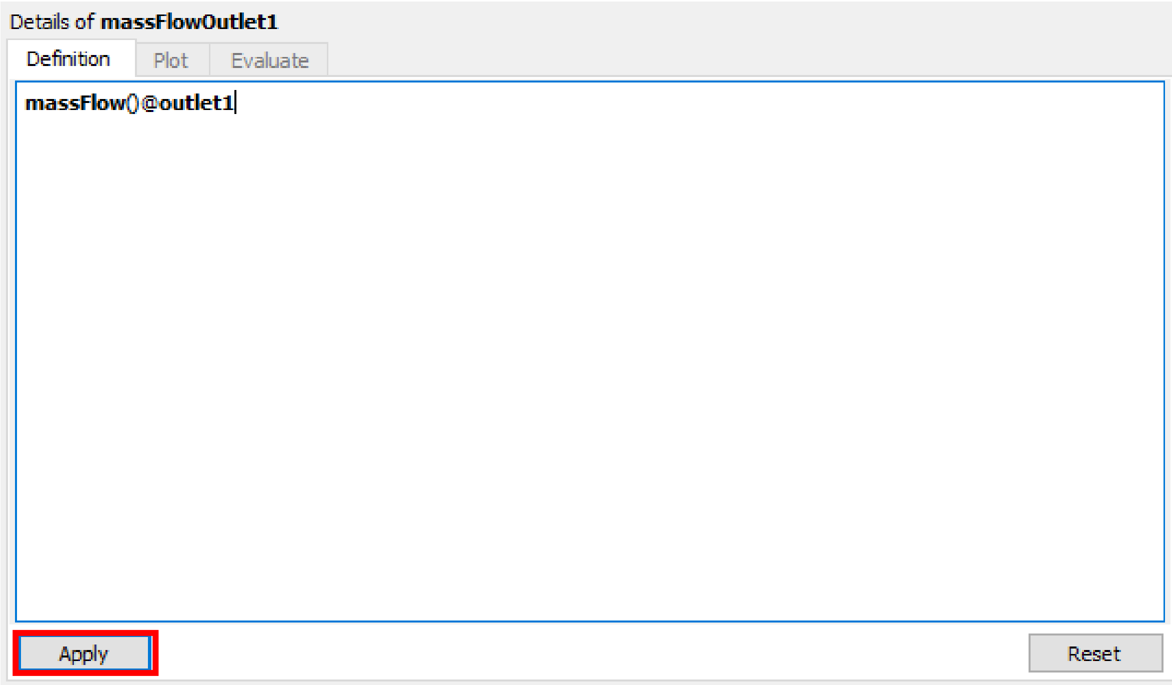
20) Apply the following definition: massFlow@inlet
Confirm *Apply*.



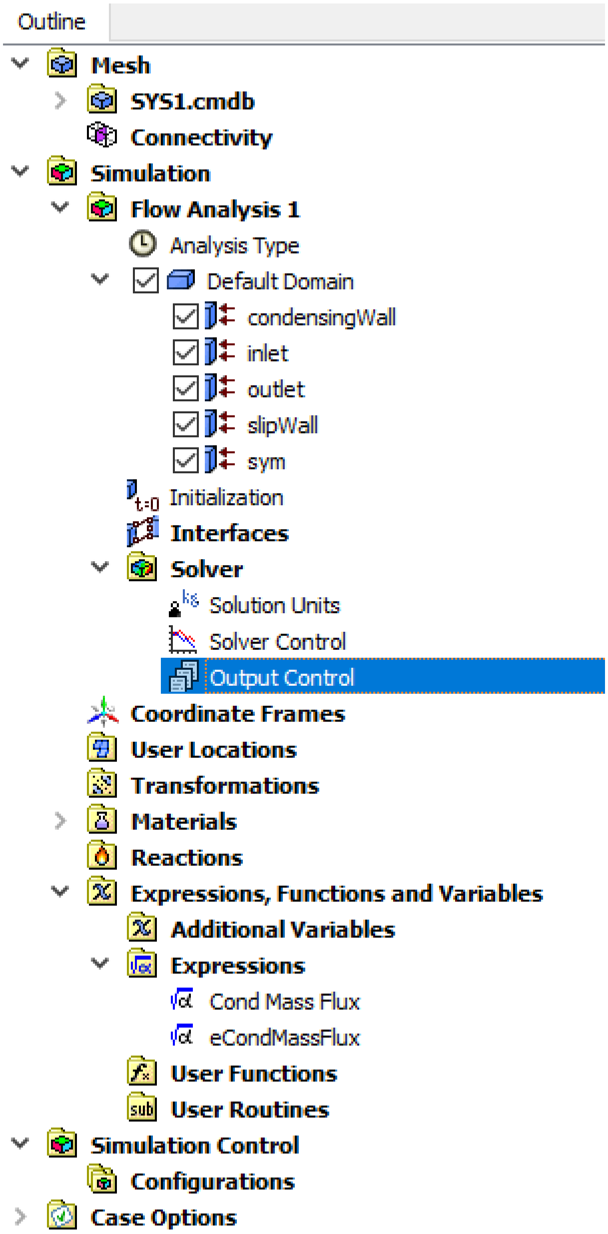
21) Create *expression* named *massFlowOutlet1*



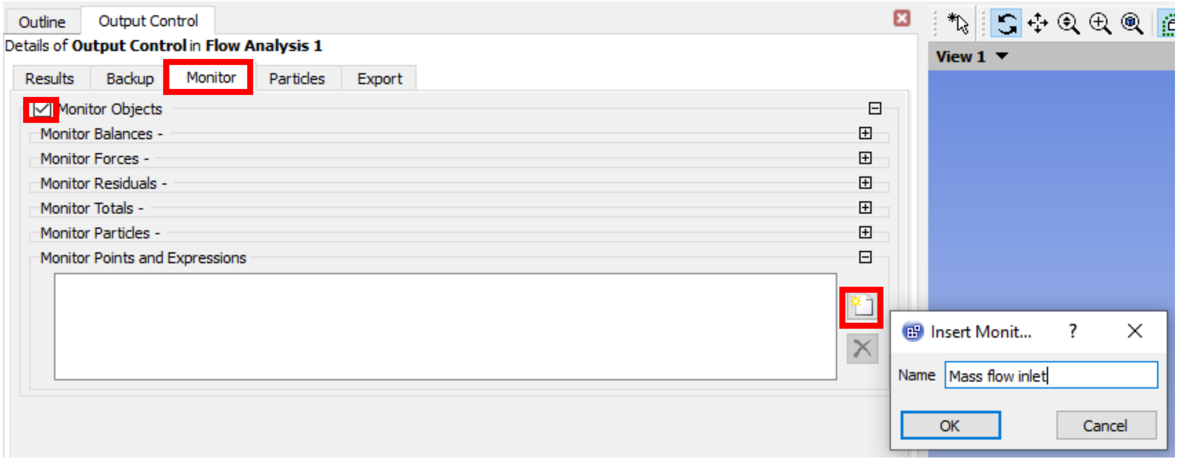
22) Apply the following definition: massFlow@outlet1
Confirm *Apply*.

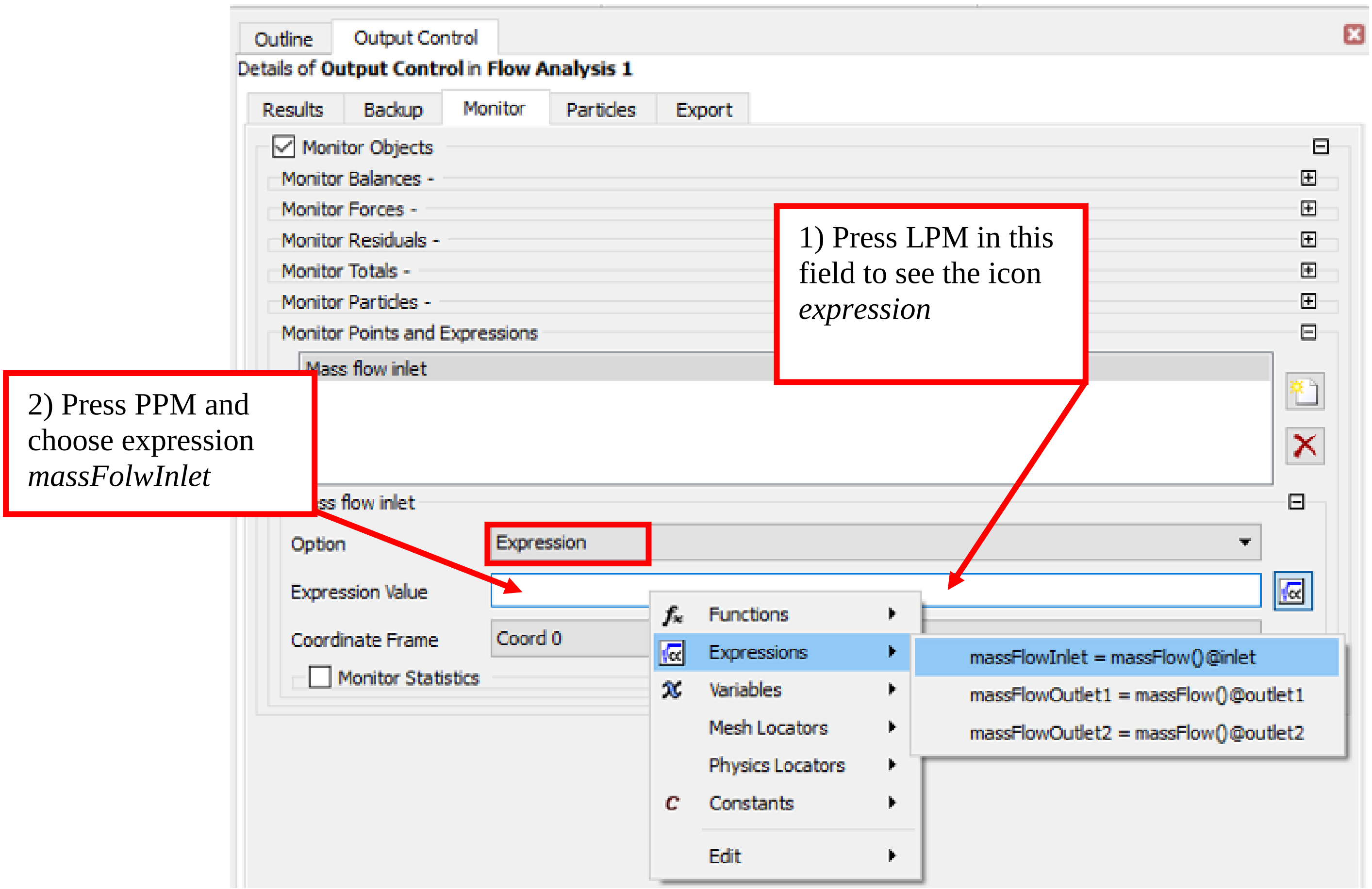


23) Create similarly *expression* named *massFlowOutlet2*
24) Open *Output Control*

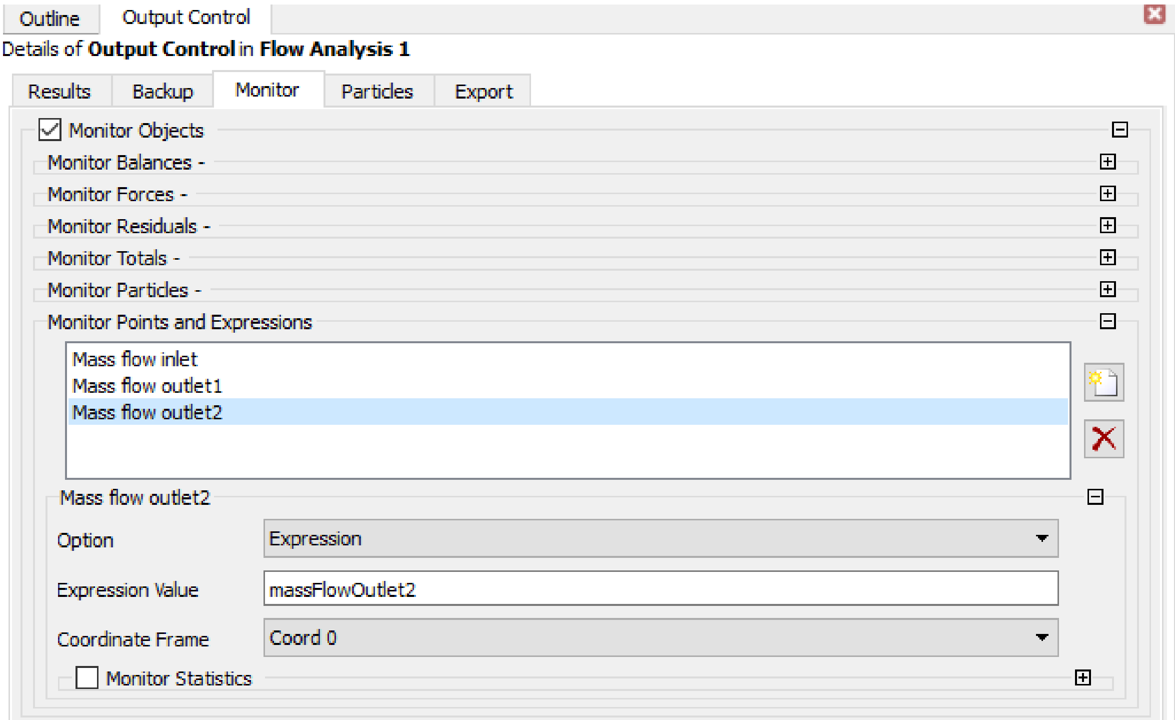


Apply the following settings and confirm *OK*.

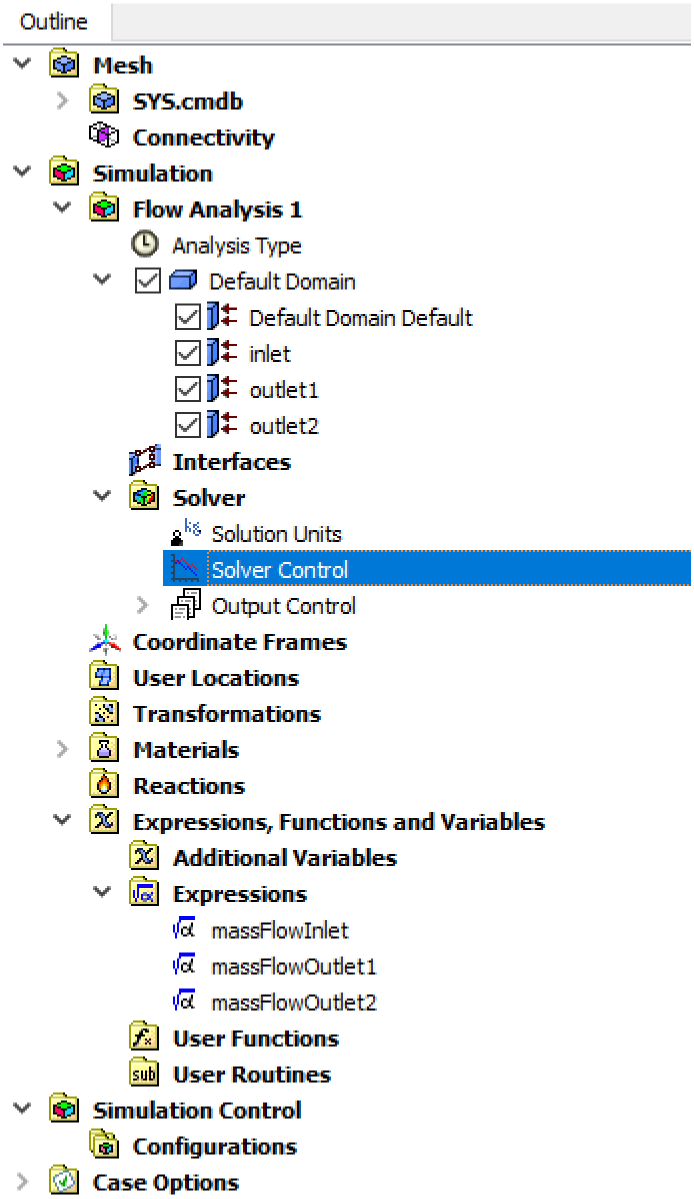


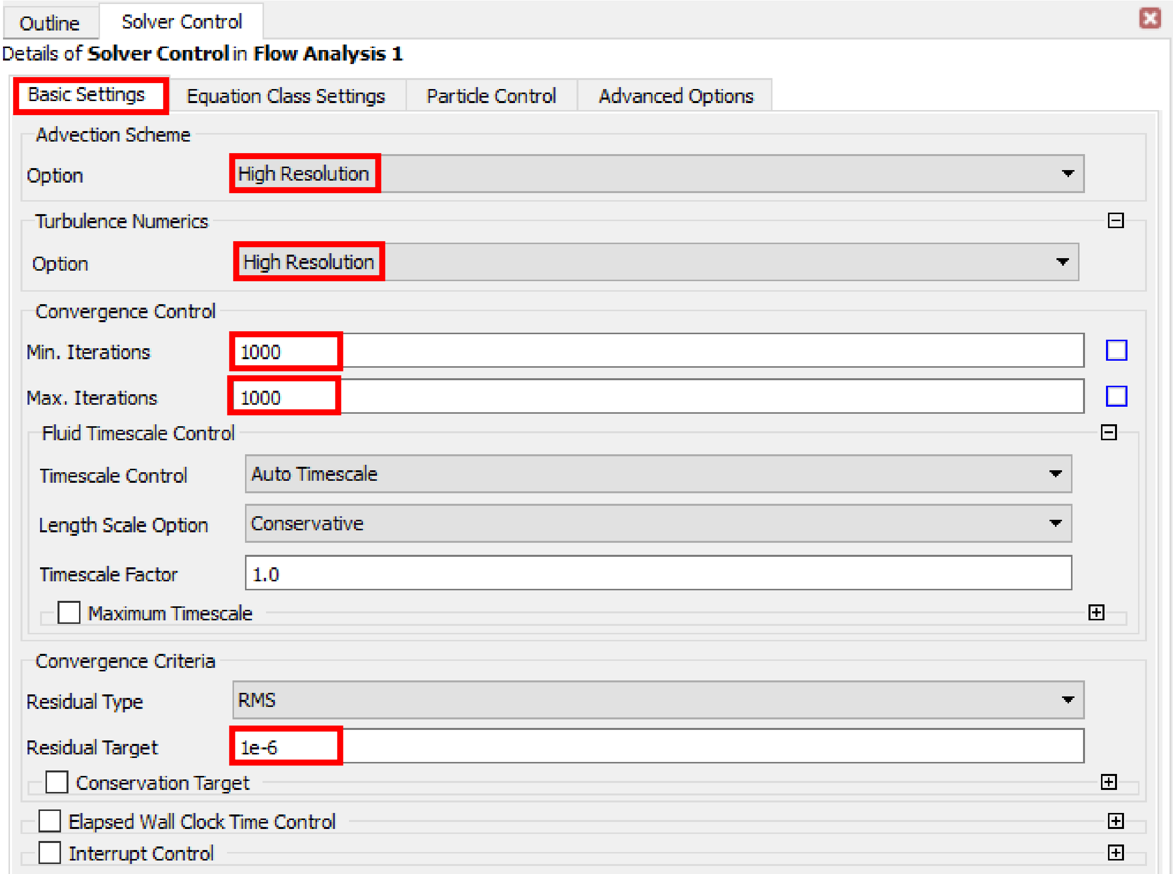


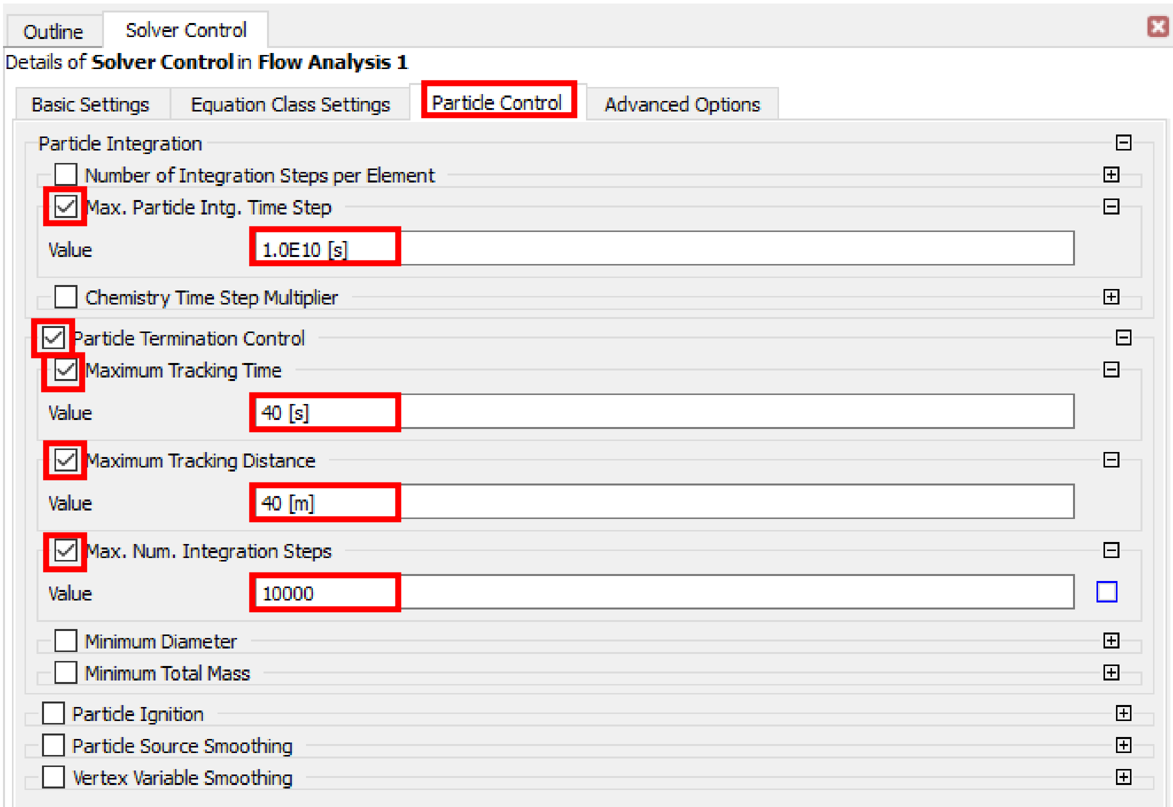
25) Similarly, create two additional *Monitor Points* using the other two expressions, and then confirm *OK*.



26) Open *Solver Control* and apply below settings


Outline
Mesh
SYS.cmdb
Connectivity
Simulation
Flow Analysis 1
Analysis Type
Default Domain
Default Domain Default
inlet
outlet1
outlet2
Interfaces
Solver
Solution Units
Solver Control
Output Control
Coordinate Frames
User Locations
Transformations
Materials
Reactions
Expressions, Functions and Variables
Additional Variables
Expressions
massFlowInlet
massFlowOutlet1
massFlowOutlet2
User Functions
User Routines
Simulation Control
Configurations
Case Options





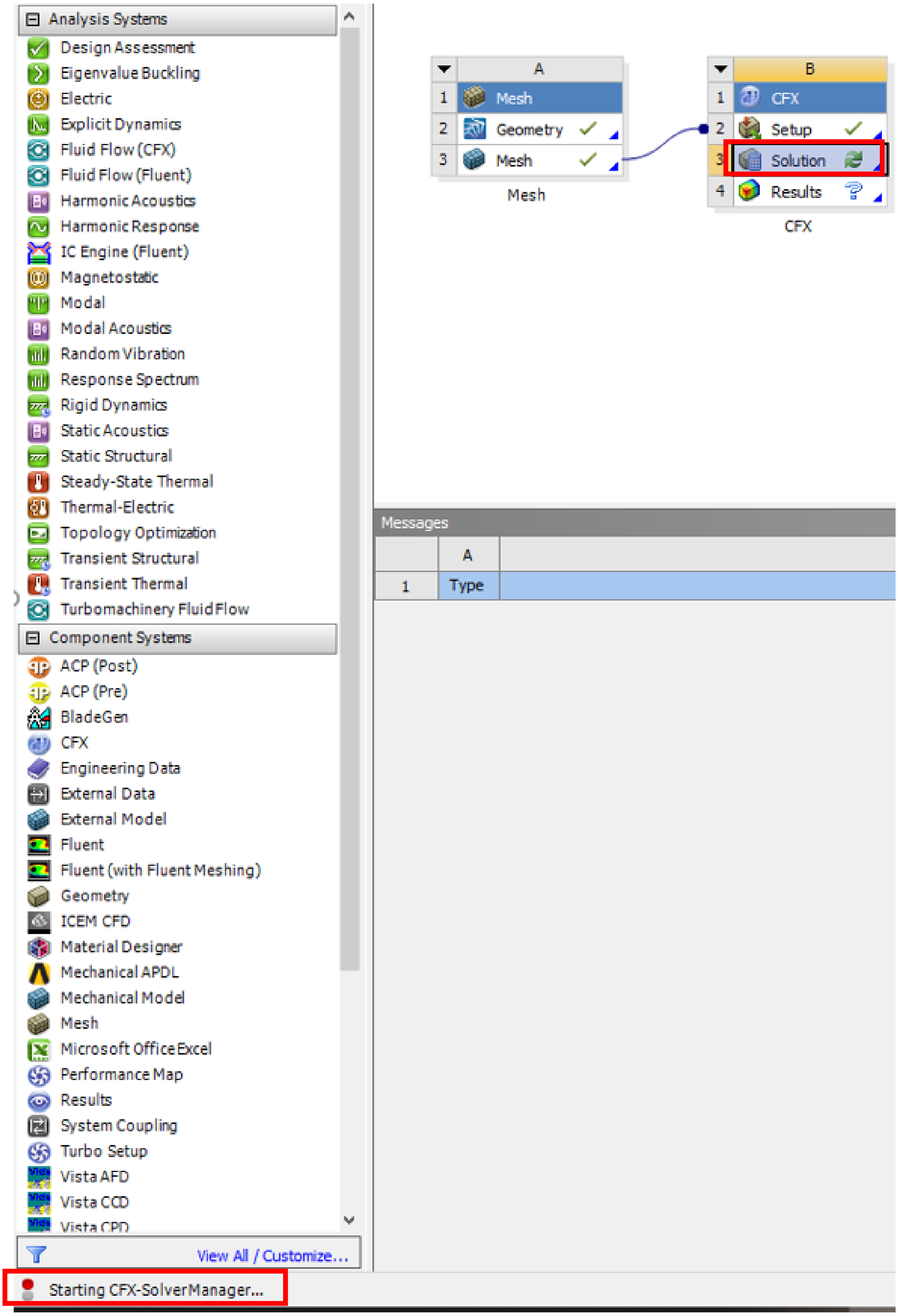
Confirm *OK*.

Value of *Max. Num. Integration Steps* controls the number of mesh elements that the particle can pass through, so you need to consider the mesh size and density when setting this parameter. All the numerical values in the table above have been designed to impose an upper limit on the amount of particle processing. For example, a particle could fall into a vortex and then be tracked all the time, which would not be necessary but would prolong the calculation. The tracking time of 40 seconds is to ensure that the particles reach the outlets. [1].

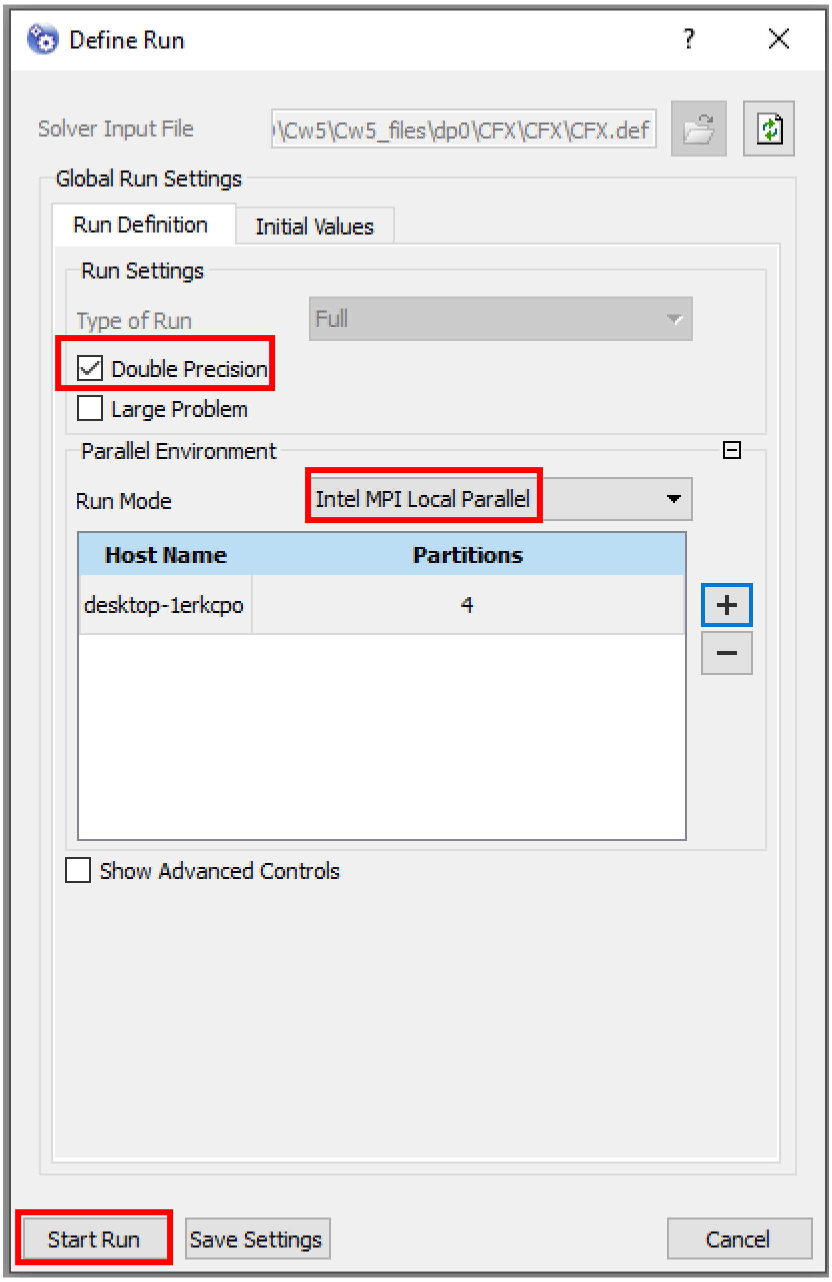
27) Close *Ansys CFX*.

2.4. CALCULATIONS

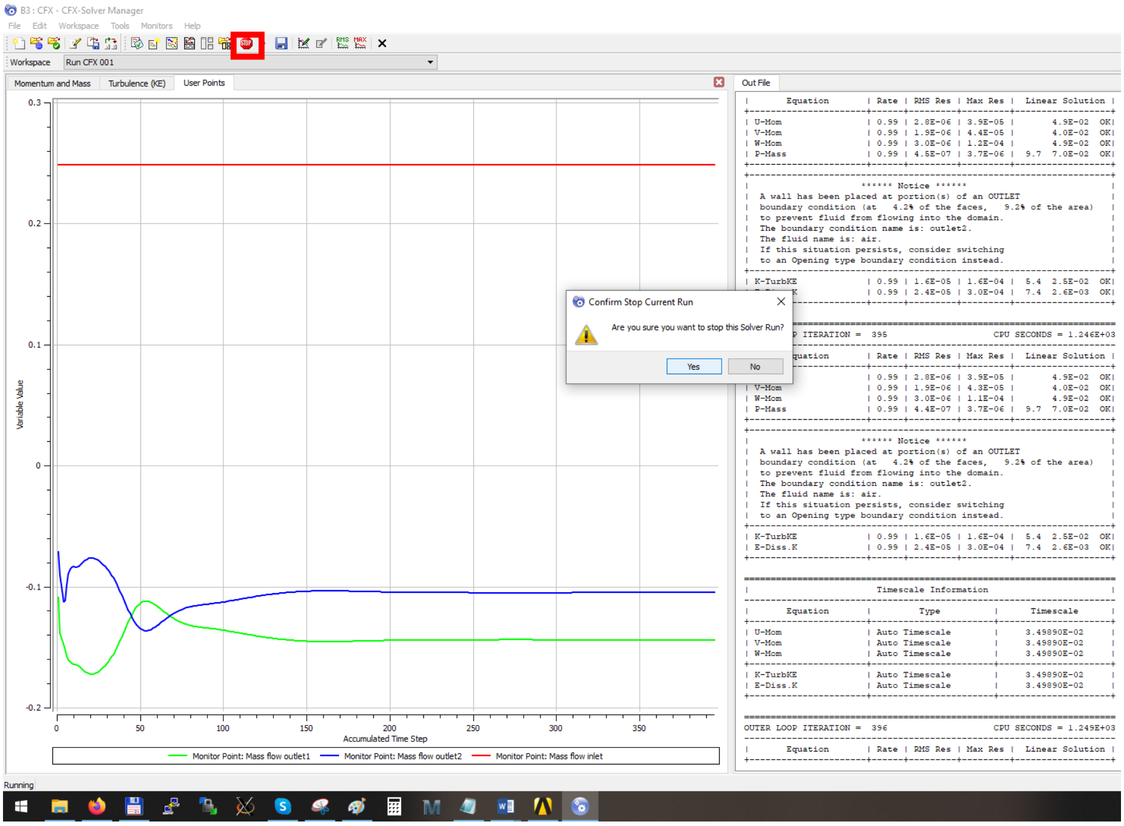
1) Double-click *Solution* to run *Ansys CFX Solver Manager*



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.



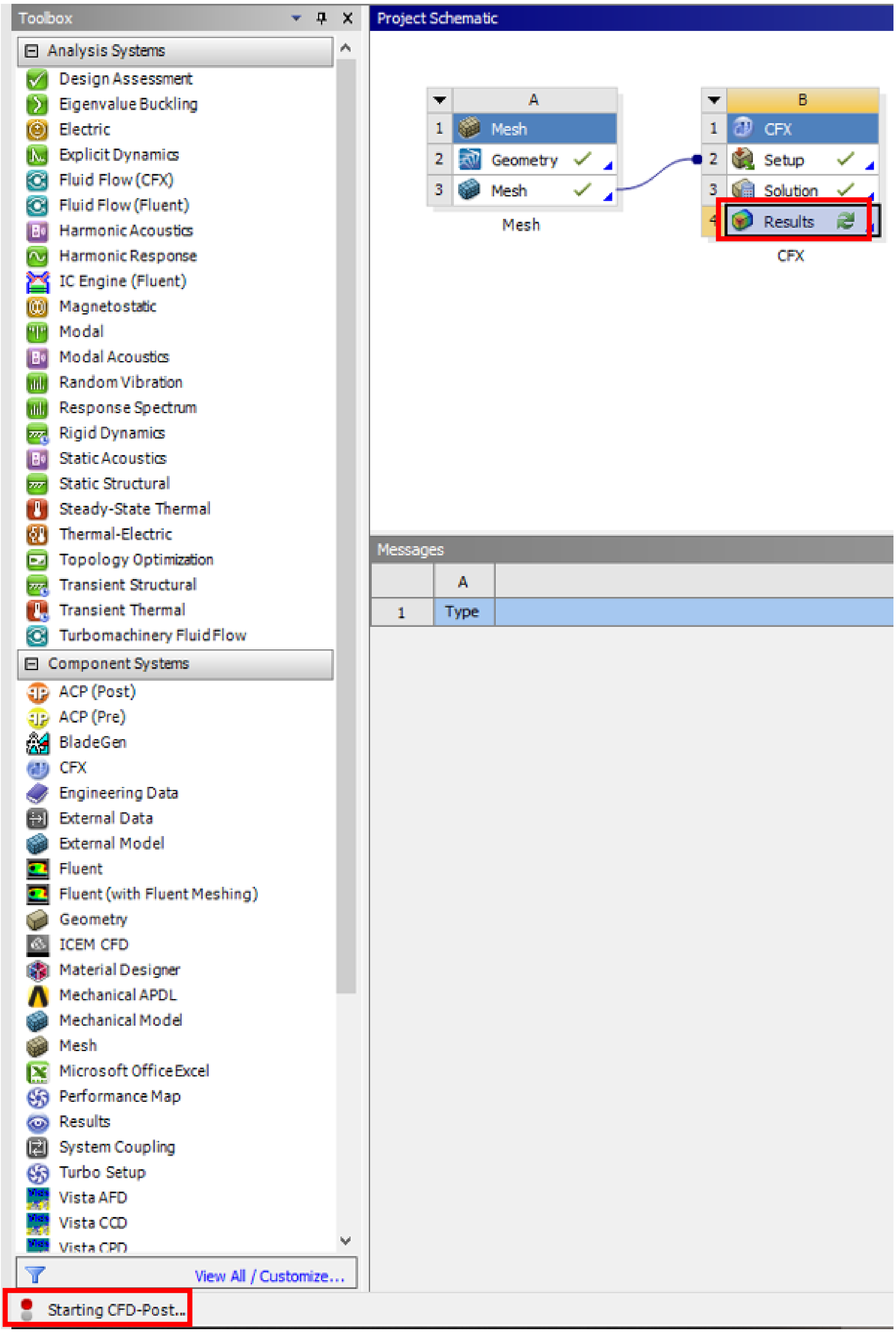
3) Calculations take about 10-15 minutes. Watch the individual bookmark tabs as they change. Pay particular attention to the *User Point* tab, where the air stream at the inlet and both outlets are shown. Steady state will be reached when the curve stabilizes, which will occur after about 350 iterations. Additional iterations should be performed until all residuals have stabilized. To stop earlier, press the *Stop* button at the top of the screen and confirm *Yes*.



4) After completing the calculations, the program will display a confirmation message.
5) Confirm *OK* and close *Ansys CFX Solver Manager*. Save project in *Workbench.*

2.5. RESULTS

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

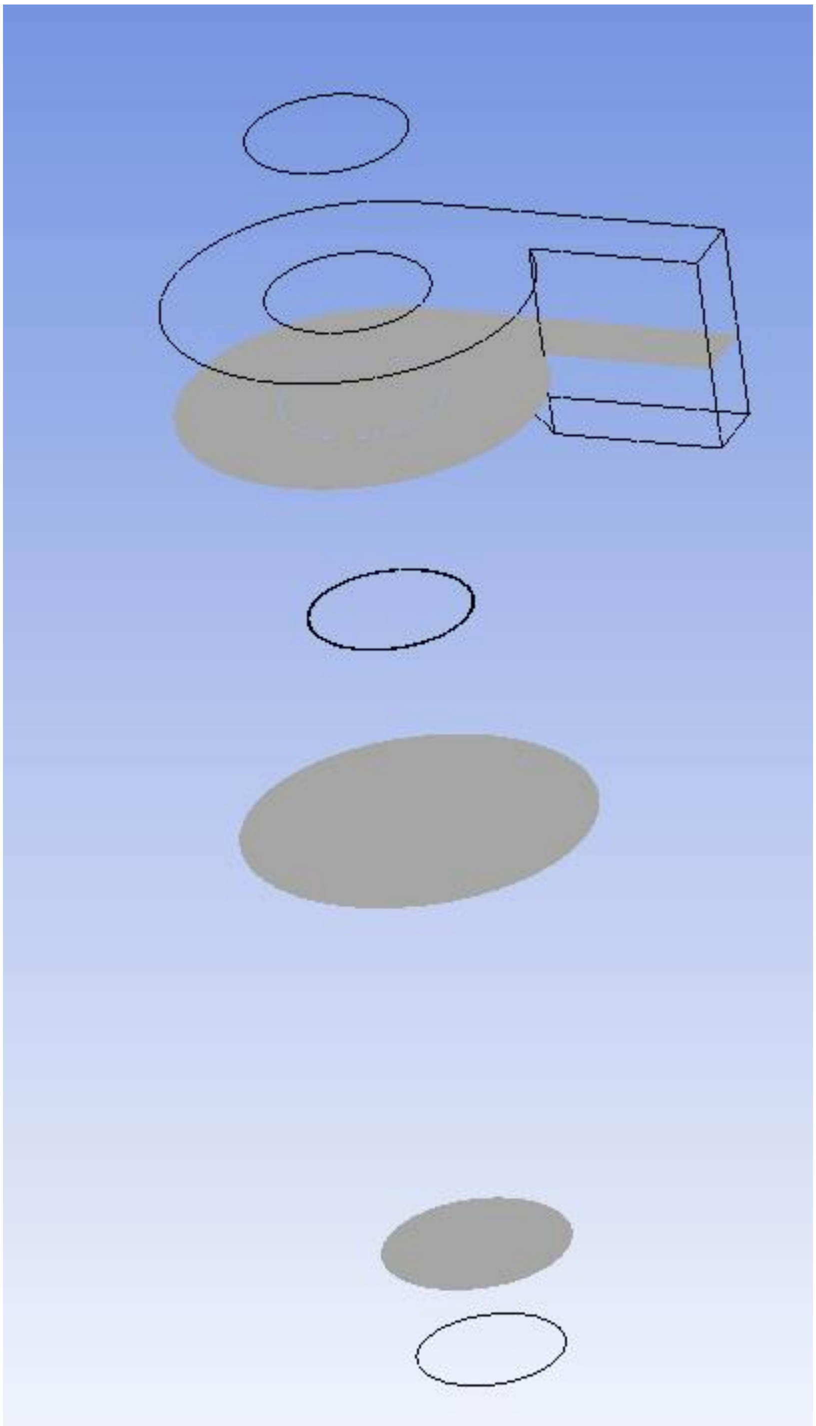


2) Create a plane passing through the cyclone axis and show the contours

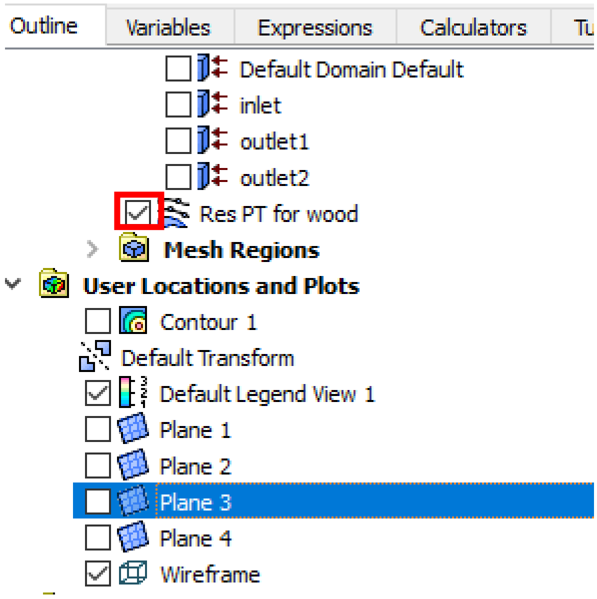
   a. *Pressure*
   b. *Velocity*
   c. *Velocity vectors*
   d. *Streamlines*

3) Create planes perpendicular to the cyclone axis for Y = 0.2, 1 and 1.8 m and show the contours

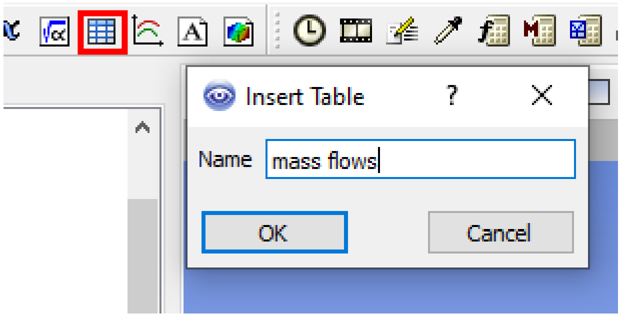
a. *Pressure*
b. *Velocity*
c. *Velocity vectors*
d. *Streamlines*



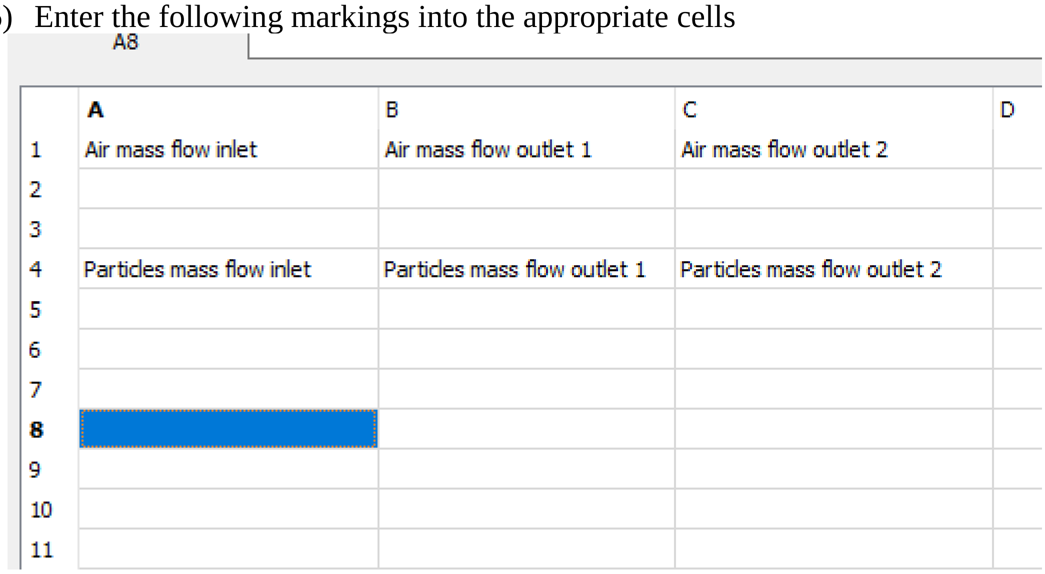
4) Save the photo of the particle trajectory by turning on the object *Res PT for wood*



5) Create table *mass flows*

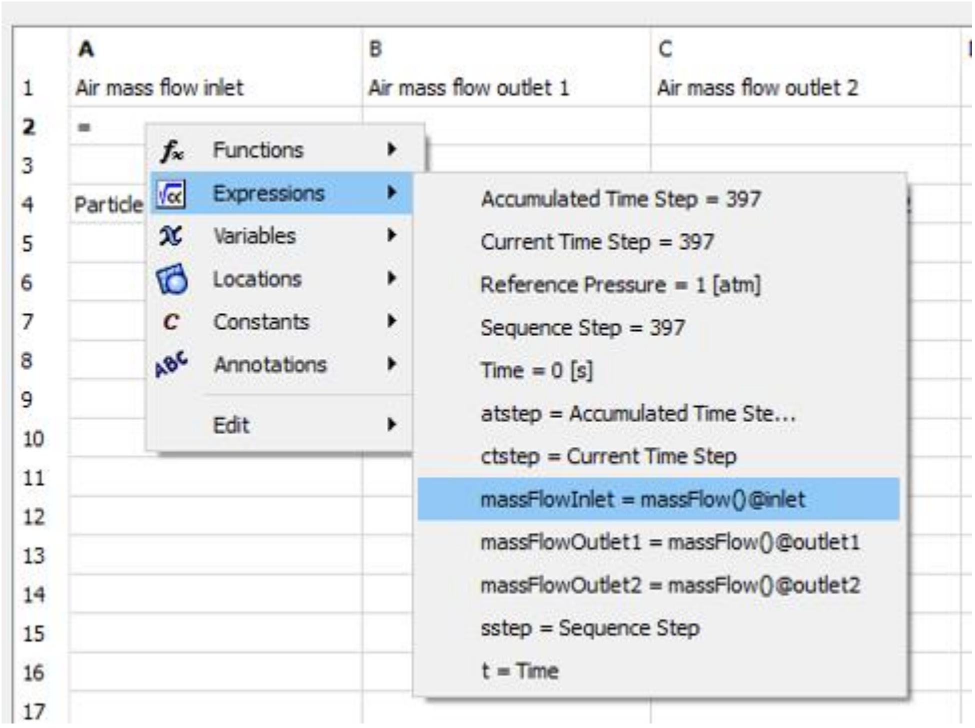


6) Enter the following markings into the appropriate cells



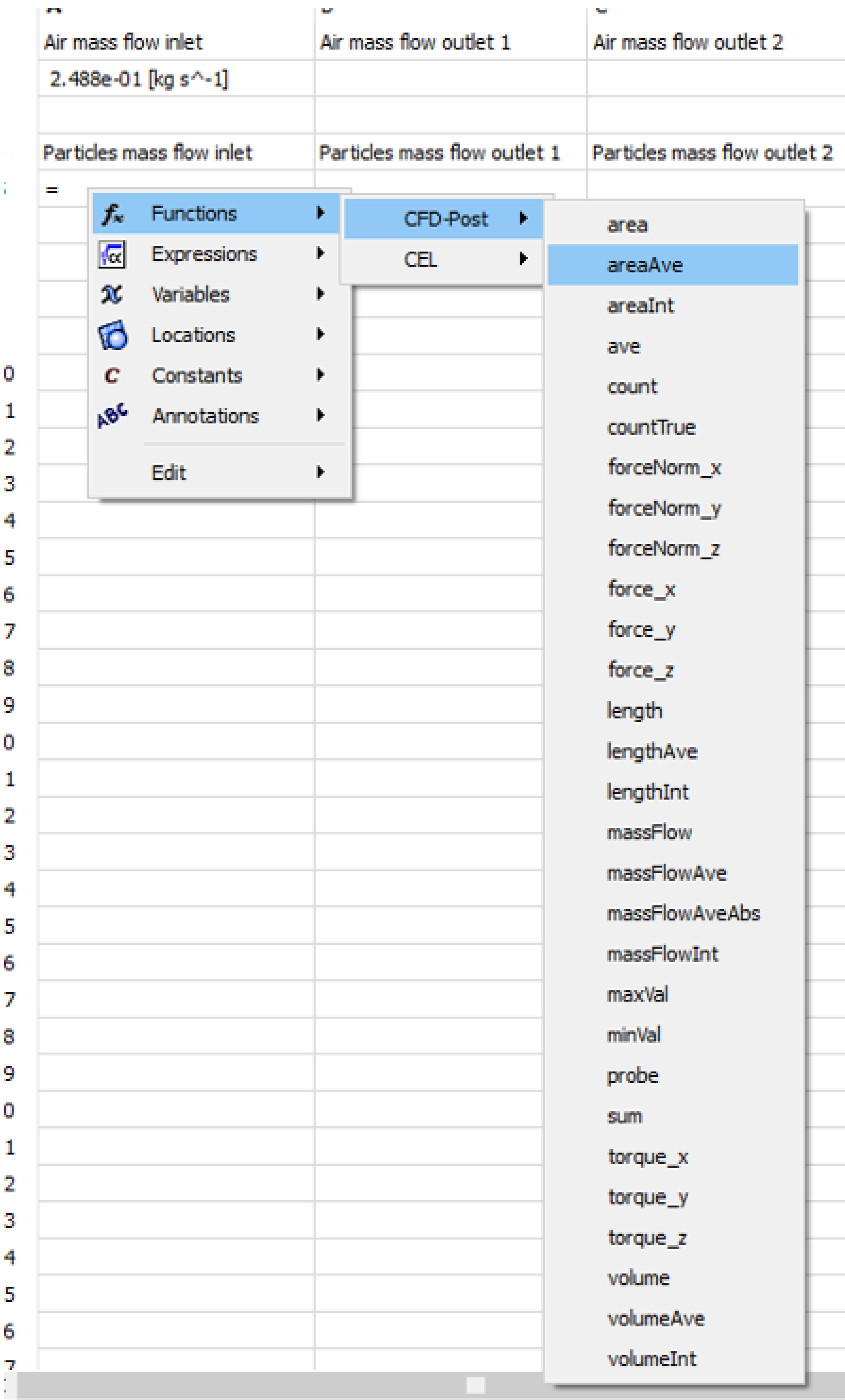
	A	B	C	D
1	Air mass flow inlet	Air mass flow outlet 1	Air mass flow outlet 2	
2				
3				
4	Particles mass flow inlet	Particles mass flow outlet 1	Particles mass flow outlet 2	
5				
6				
7				
8				
9				
10				
11				

7) Fields A2, B2 and C2 fill in with the expressions created in points 19-23. To do this, for example, in cell A2, place the cursor in A2, enter the equal sign "=", press RMB under *Expressions* select the appropriate expression

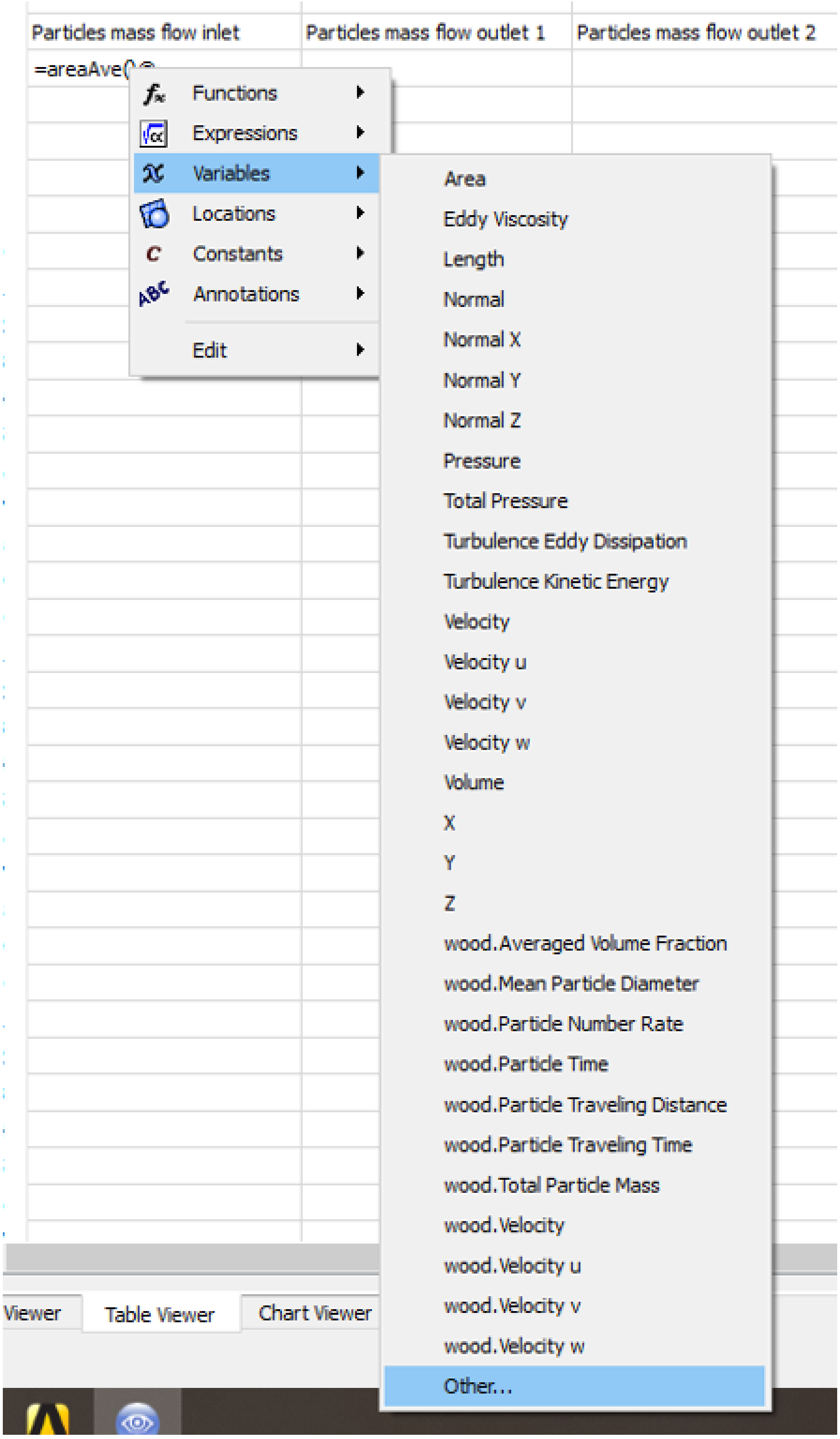


Do the same with cells B2 i C2.

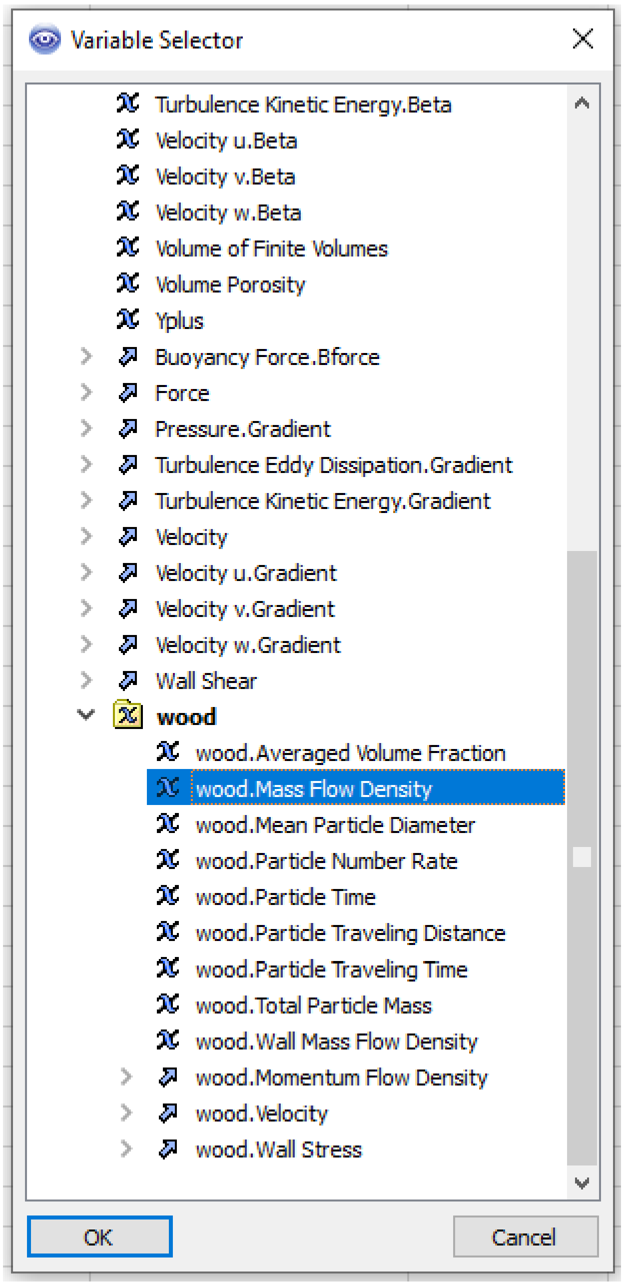
8) To calculate the particle mass flow in field A5, do the following:
   a. Type „=" in A5
   b. Press RMB and select *Functions->CFD-Post->areaAve*



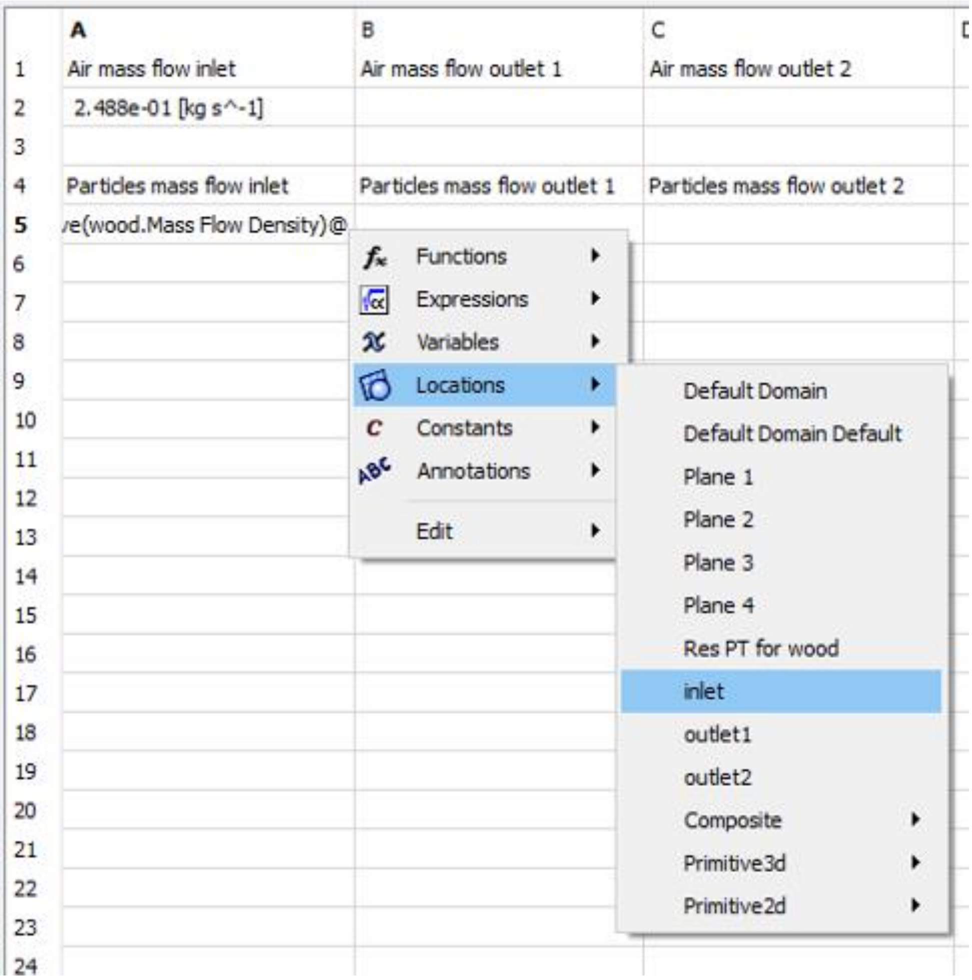
c. Position the cursor between the parentheses and press RMB. Select *Variables*-> *Other* to view all variables



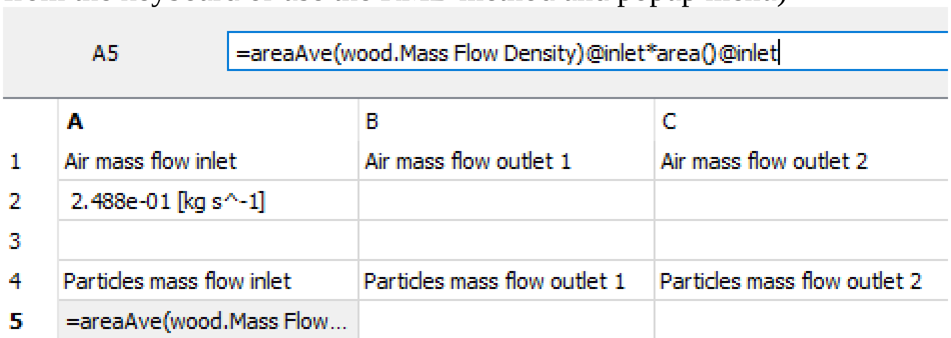
d. Select *wood.Mass Flow Density* variable and confirm *OK*



e. Position the cursor after the „@" and select *Locations->inlet*



f. Position the cursor after the expression *=areaAve(wood.Mass Flow Density)@inlet* and type „*area()@inlet" (you can enter it manually from the keyboard or use the RMB method and popup menu)



g. Press *Enter* to calculate the inlet mass flow rate

9) Perform the same actions as in point 8 for cells B5 i C5

3. RESULTS TO BE INCLUDED IN THE REPORT

1) In the plane passing through the cyclone axis and show the contours
   a. *Pressure*
   b. *Velocity*
   c. *Velocity vectors*
   d. *Streamlines*

2) In planes perpendicular to the cyclone axis for Y = 0.2, 1.0 and 1.8 m and show the contours
    a. *Pressure*
    b. *Velocity*
    c. *Velocity vectors*
    d. *Streamlines*
3) Photograph of the particle trajectory. Explain the reasons for the shape and length of the particle trajectory.
4) Streams of air and particle masses at the inlet and both outlets.

4. Optional tasks (not obligatory)

1. Perform calculations for particles with the following parameters: minimum diameter 10 µm, maximum diameter 200 µm, average diameter 100 µm, standard deviation 30 µm. Compare particle trajectories.
2. For the particles from point 1 make calculations for the *Fully coupled* model and compare both solutions.
3. Check the effect of pipe length inside the cyclone (Fig. 1) on air and particulate distribution. To do this, edit the geometry by shortening the length of the pipe. Then update the numerical mesh and re-calculate for point 1 particles.

5. References

[1] Ansys CFX Tutorials, v 15.0, 2013.