Politechnika Wrocławska

Wydział Mechaniczno-Energetyczny
Full-time studies

Selected problems of thermal-flow processes

Exercise no. 2

Transient heat transfer with taking into account radiation heat transfer

Wrocław 2020

TABLE OF CONTENTS

1. INTRODUCTION

The exercise will show how to model transient heat transfer including radiation. As a starting point, the model from *Exercise no. 1* will be used, which will be extended to a two-dimensional case to take into account convective heat exchange on the outer surface of the pipe. In the final stage, the radiation component of the heat stream from the outer surface of the pipe will also be taken into account. The diagram of the analyzed case is presented in Fig. 1.

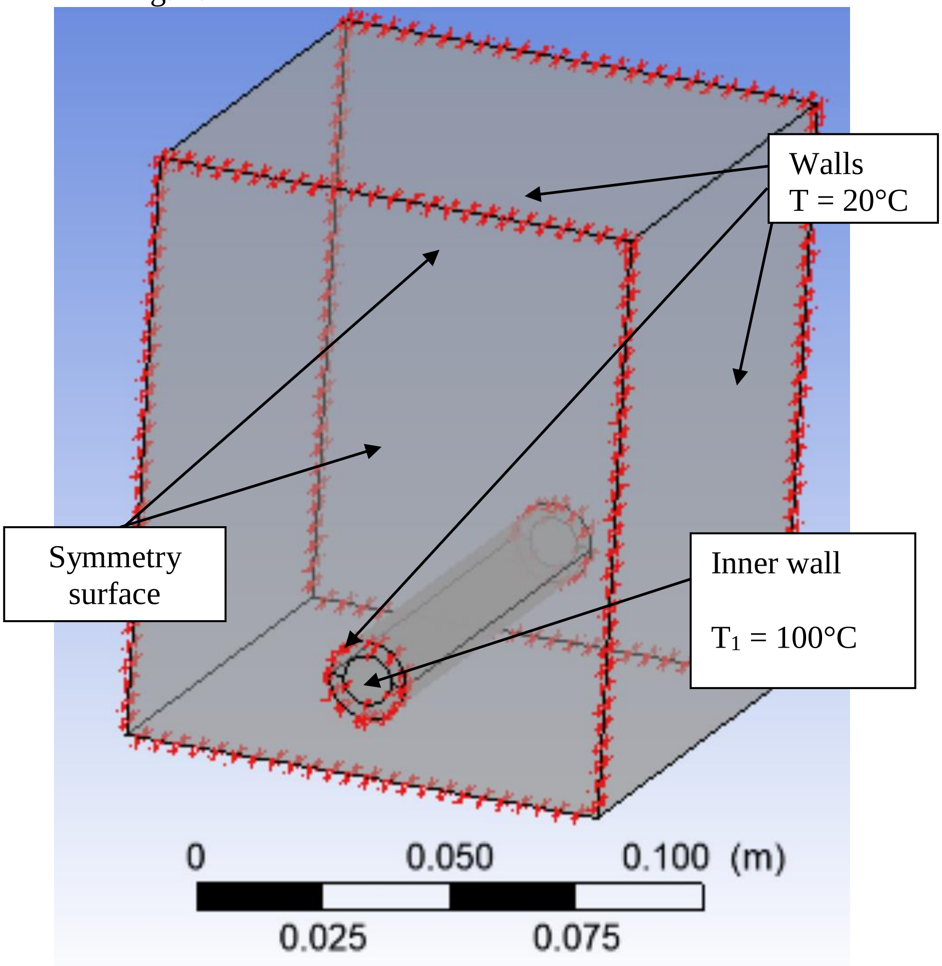


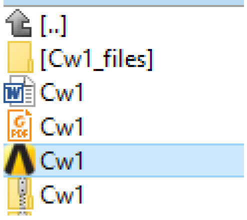
Fig. 1. Diagram of the issue of complex heat exchange

2. One-dimensional transient heat conduction through a cylindrical wall with taking into account convection and radiation

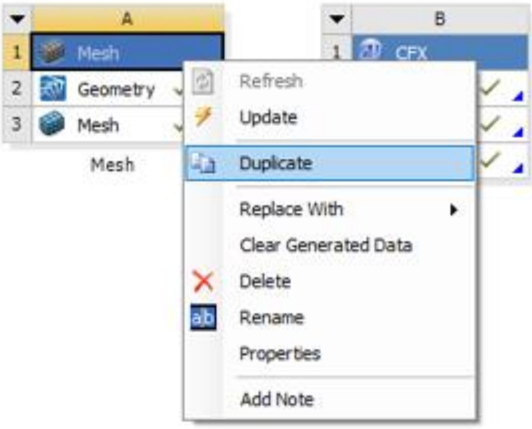
2.1. Geometry

Do the following:

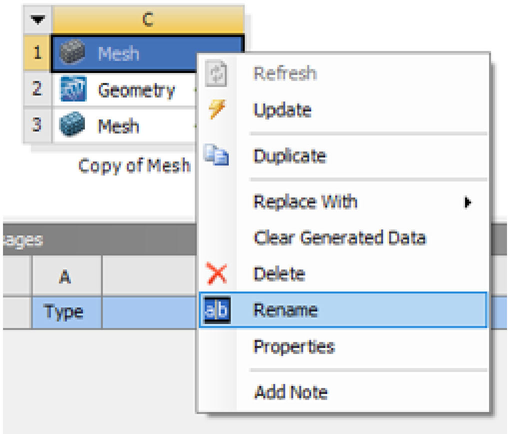
1) Open *Workbench* file containing *Exercise no. 1*



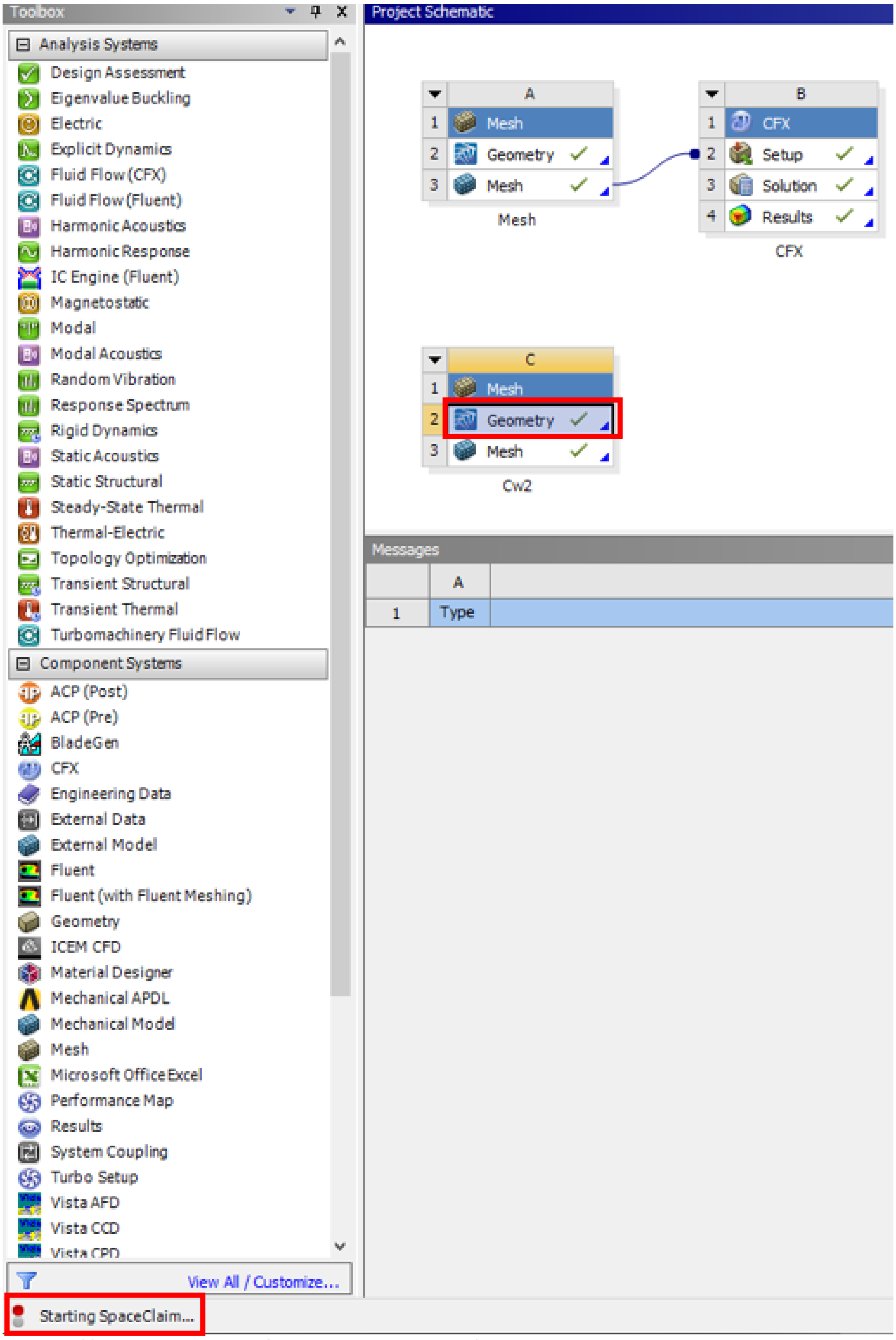
2) RMB click on *Mesh* and select *Duplicate* to copy geometry from *Exercise no. 1*



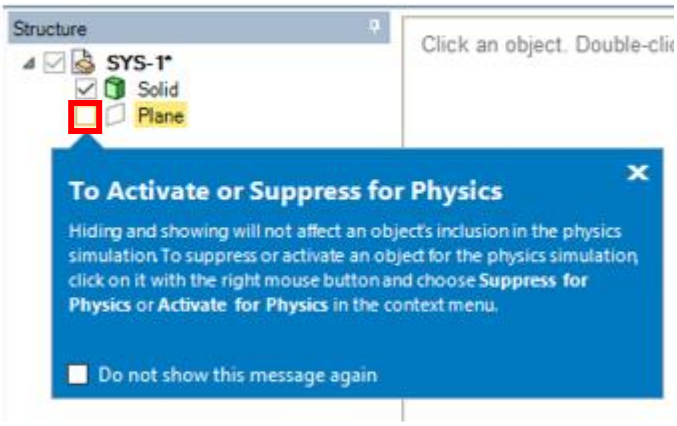
3) RMB click on *Mesh* and select *Rename*. Change name into *Cw2*



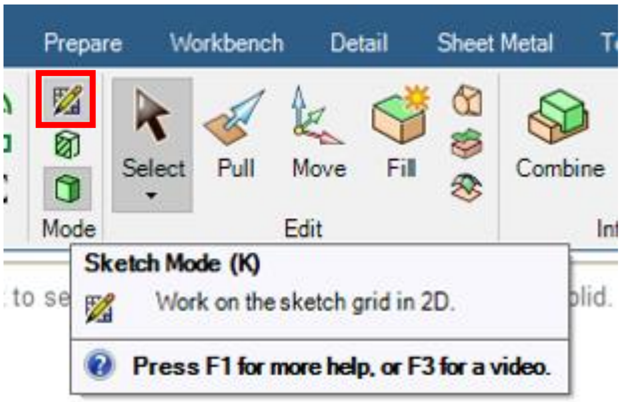
4) Double click LMB on *Geometry* in *Cw2*



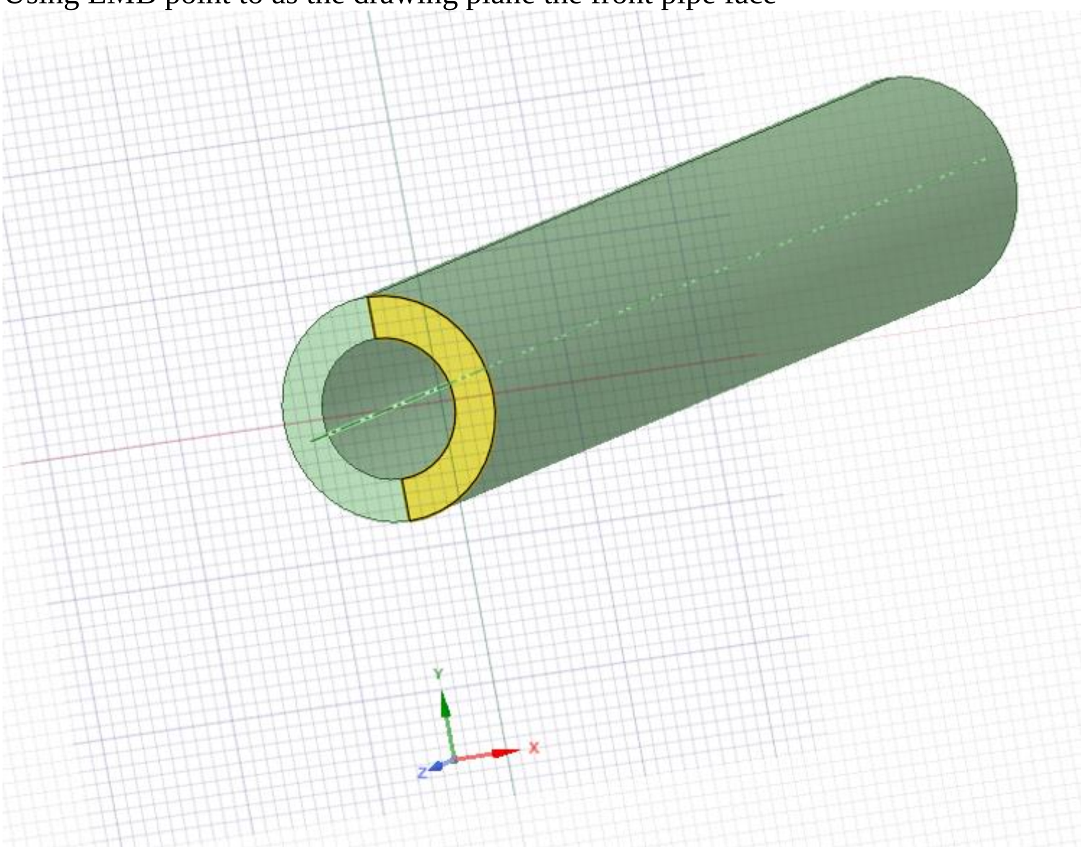
5) Turn off the visibility of the cutting plane from *Exercise no. 1*

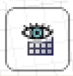


6) Choose *Sketch Mode*

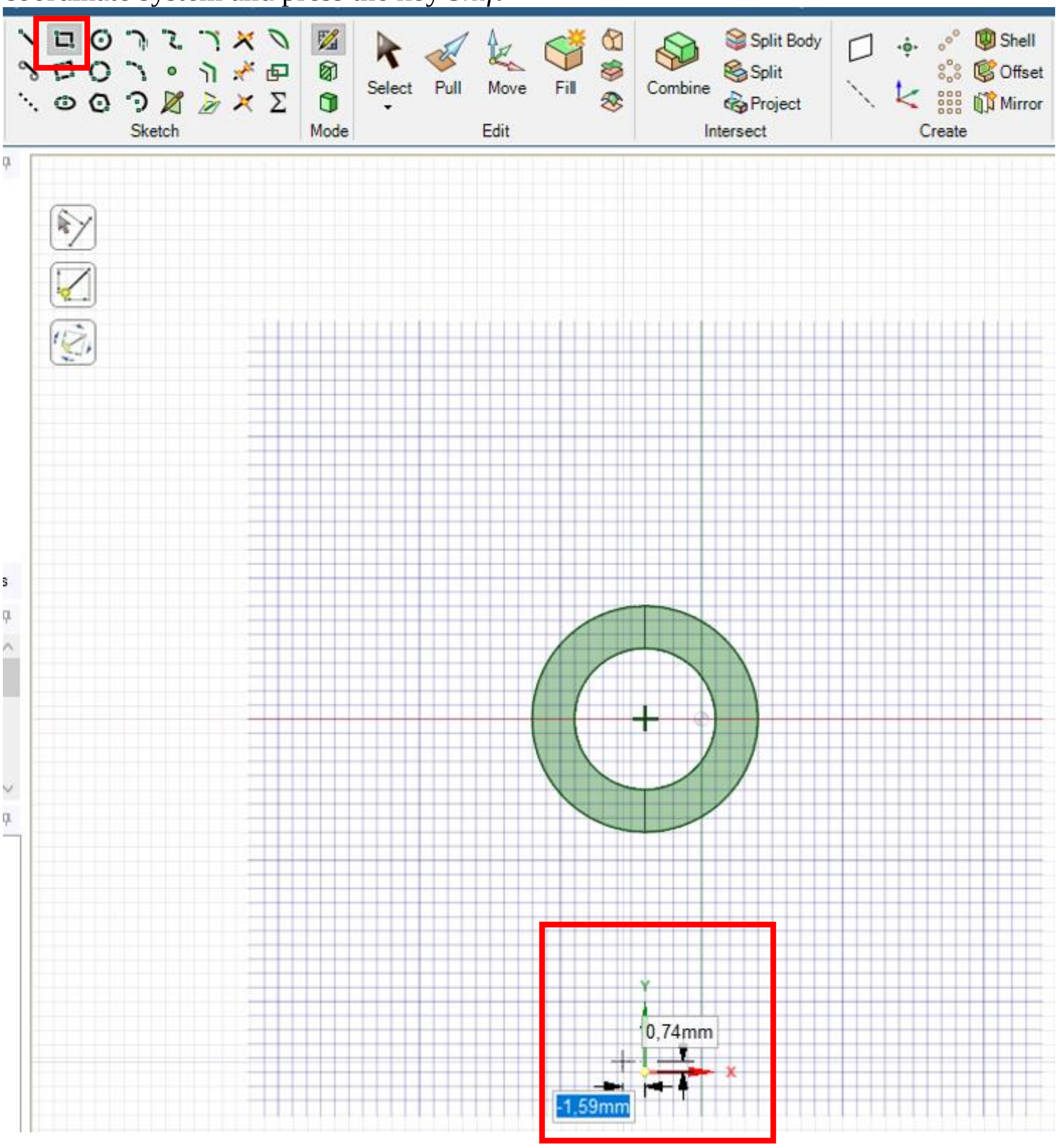


7) Using LMB point to as the drawing plane the front pipe face

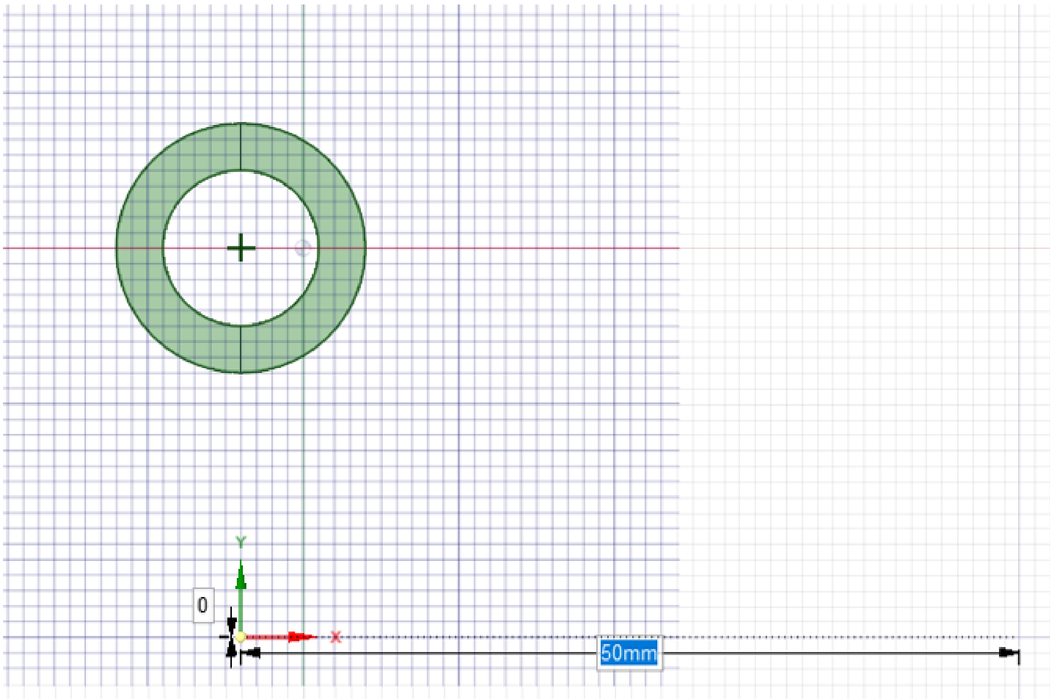


8) Set the view parallel to the screen (*Shift* + *v* lub )

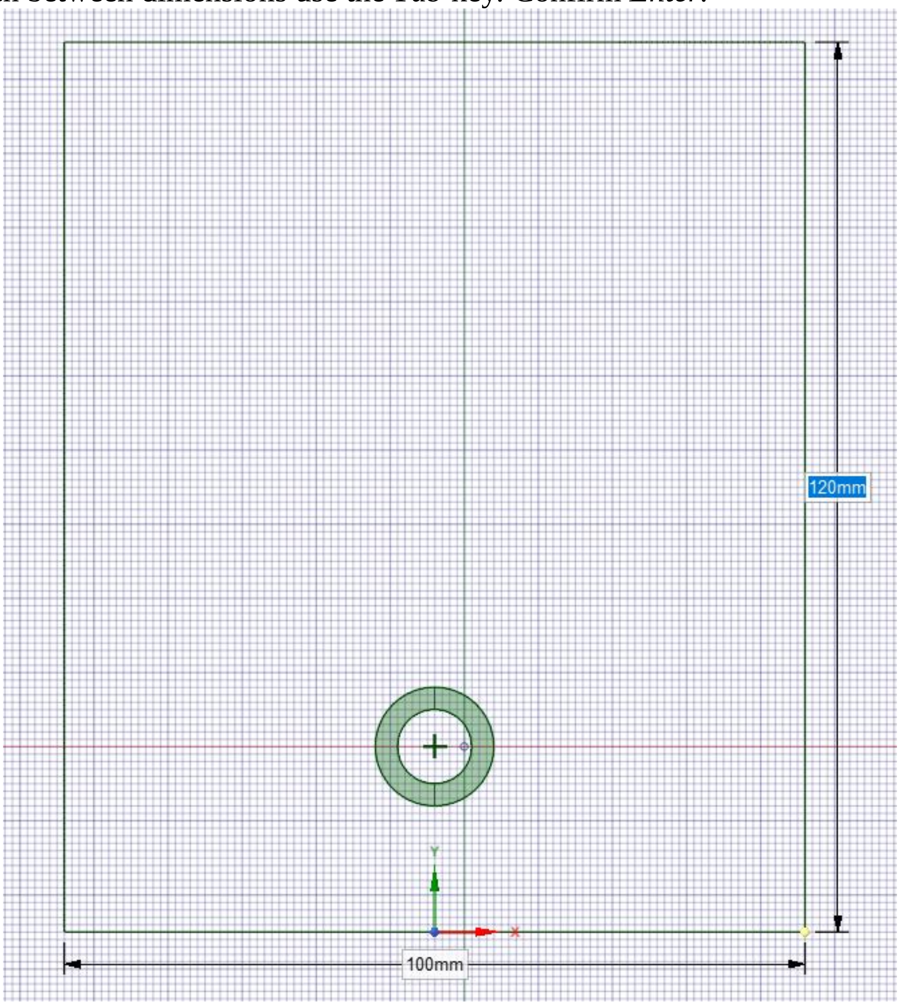
9) Select the rectangle drawing icon, move the cursor to the center of the coordinate system and press the key *Shift*



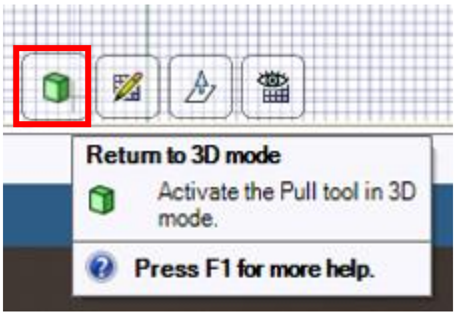
10) After *Shift* clicking two dimension edit fields will appear. Set horizontal dimension as 50 mm and vertical dimension as 0 mm. To switch between dimensions use the *Tab* key. Confirm *Enter*.



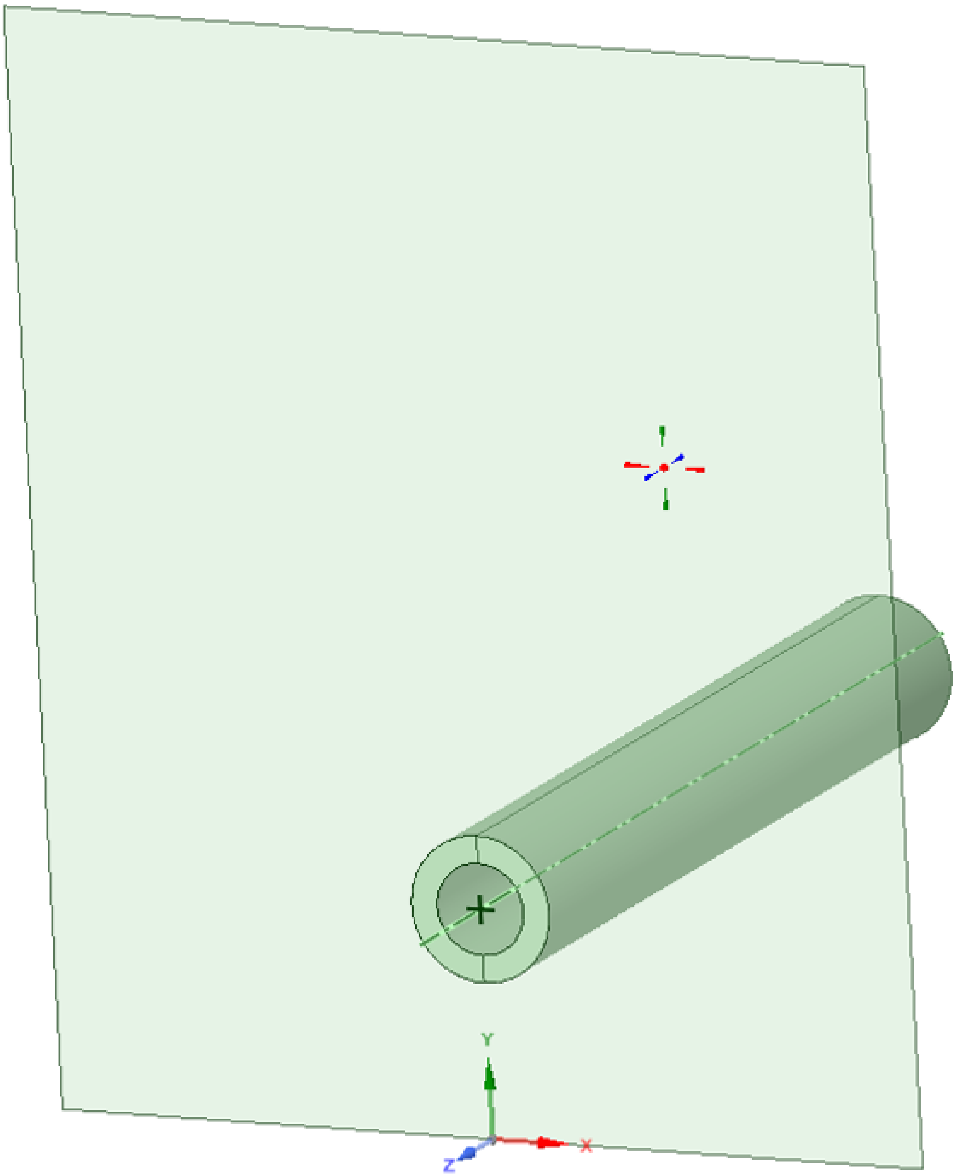
11) Set horizontal dimension to 100 mm and vertical dimension as 120 mm. To switch between dimensions use the *Tab* key. Confirm *Enter*.



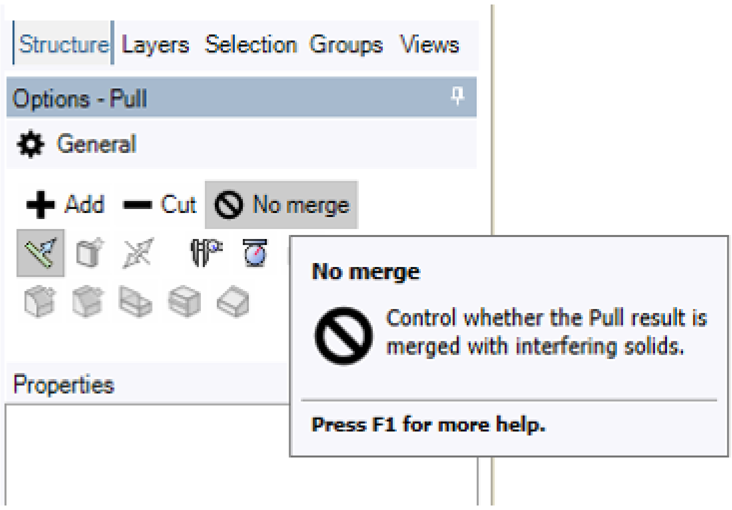
12) Return to 3D view



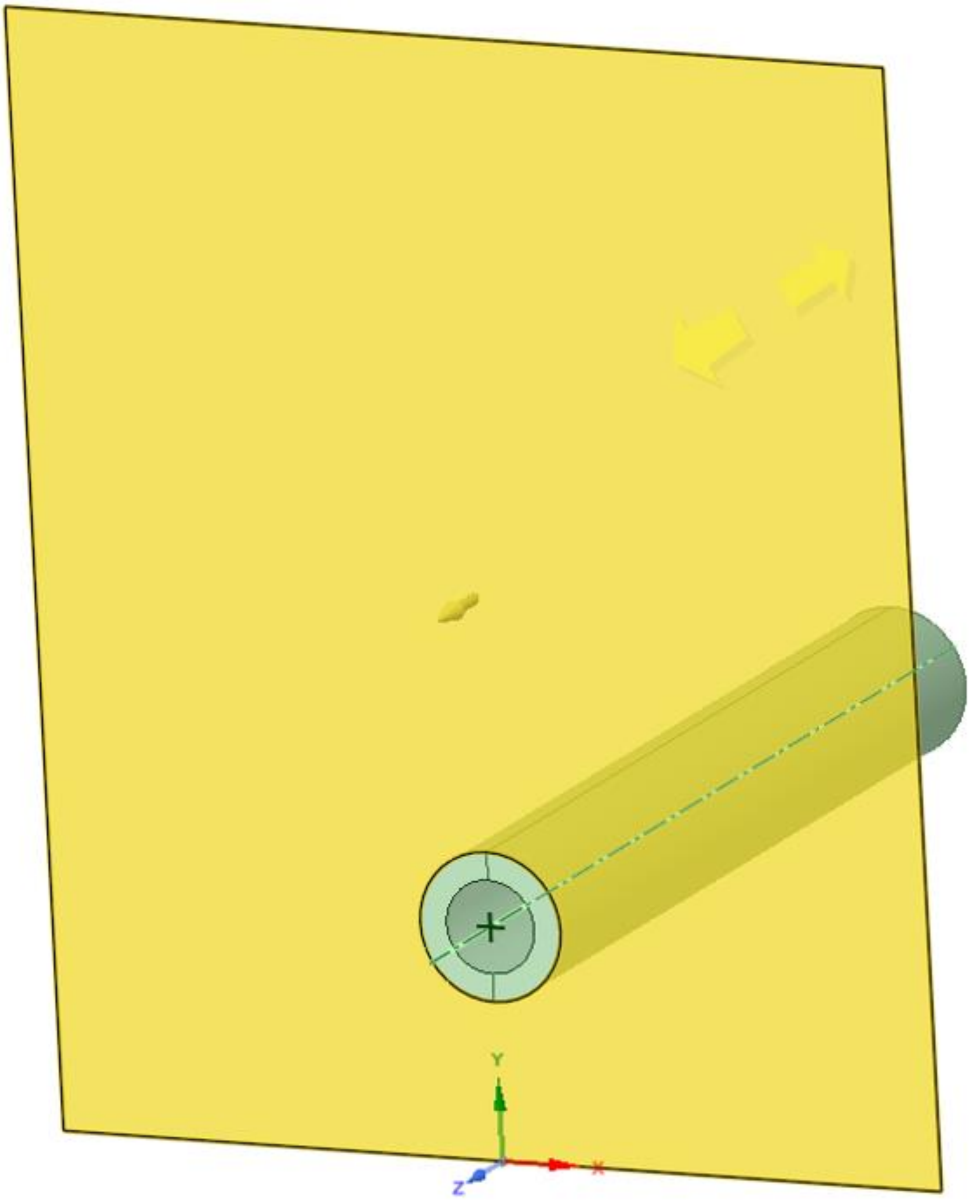
13) By sliding the *Scroll* button pressed rotate the view as below



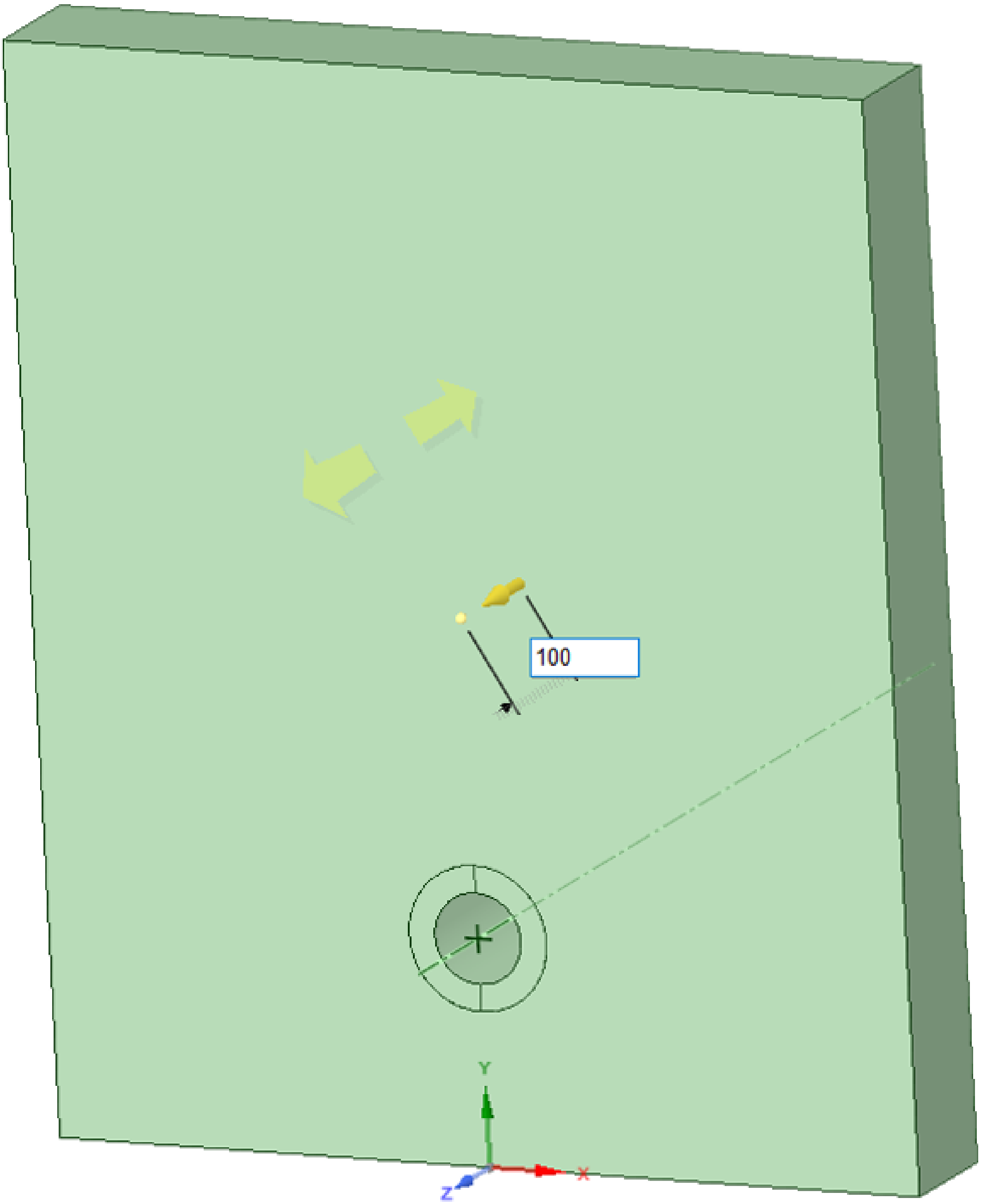
14) On the left side of the screen, with LMB select *No merge*



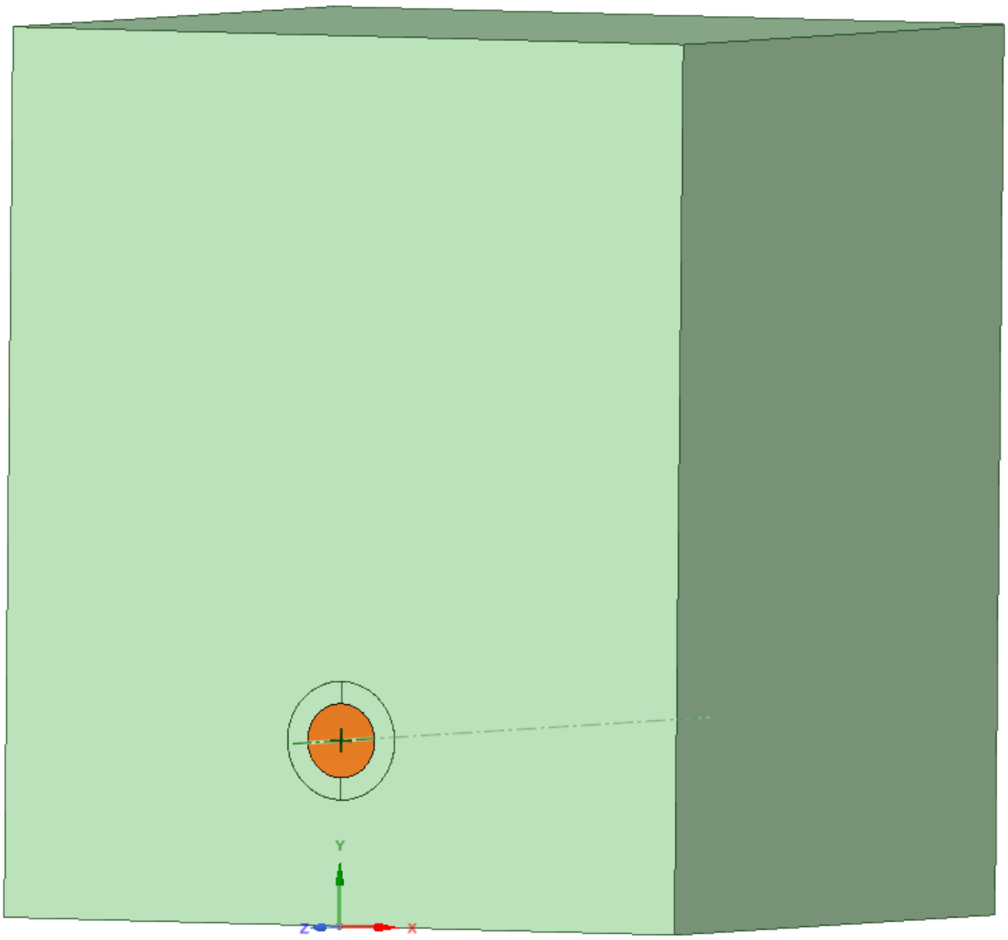
15) Then position the cursor so that the yellow arrows show up



16) Move the cursor while LMB is pressed and set the value to 100 mm, then confirm *Enter*.



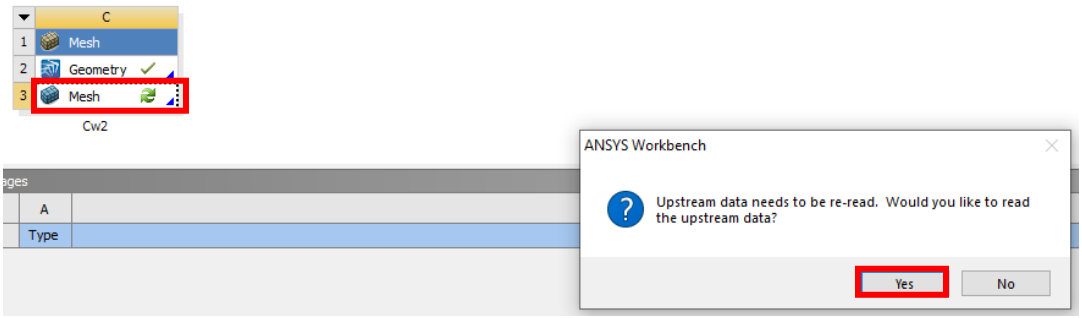
17) Press *Esc* to go out from *Pull* and select with LMB pipe inlet surface



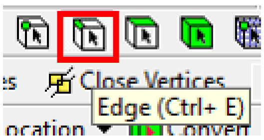
18) Press *Delete* to remove the orange surface and close *Spaceclaim.*
19) Save project in *Workbench* (*Ctrl + s*).

2.2. Numerical mesh

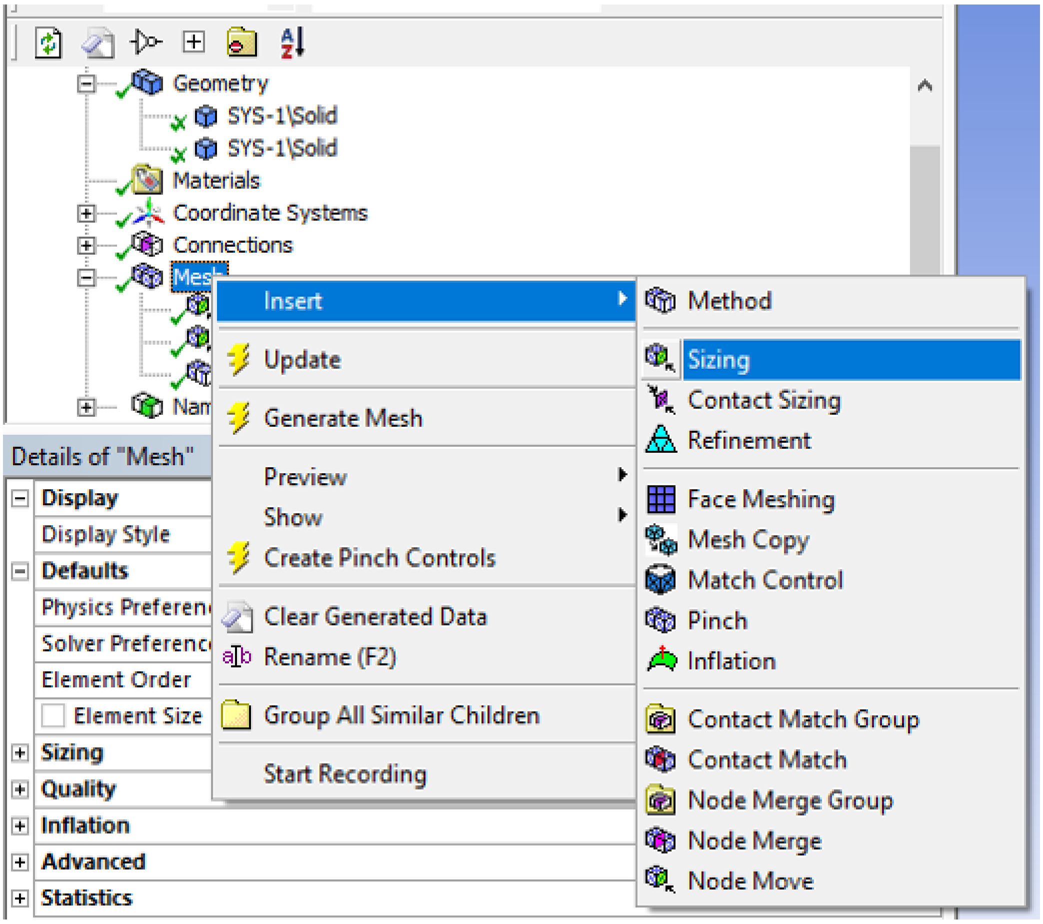
1) Double click on *Mesh* and select *Yes*



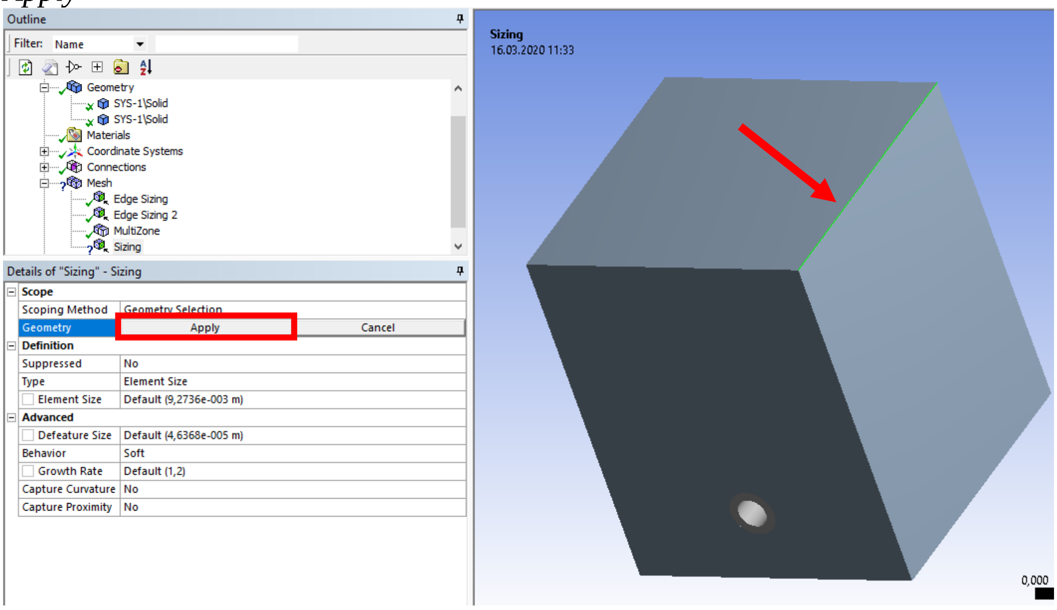
2) Select an edge selection filter



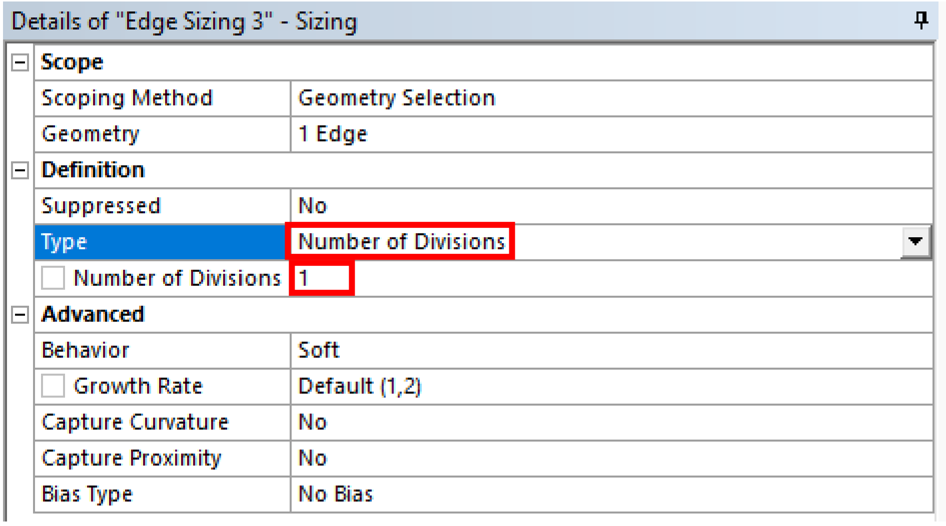
3) RMB on *Mesh* and choose *Insert->Sizing*



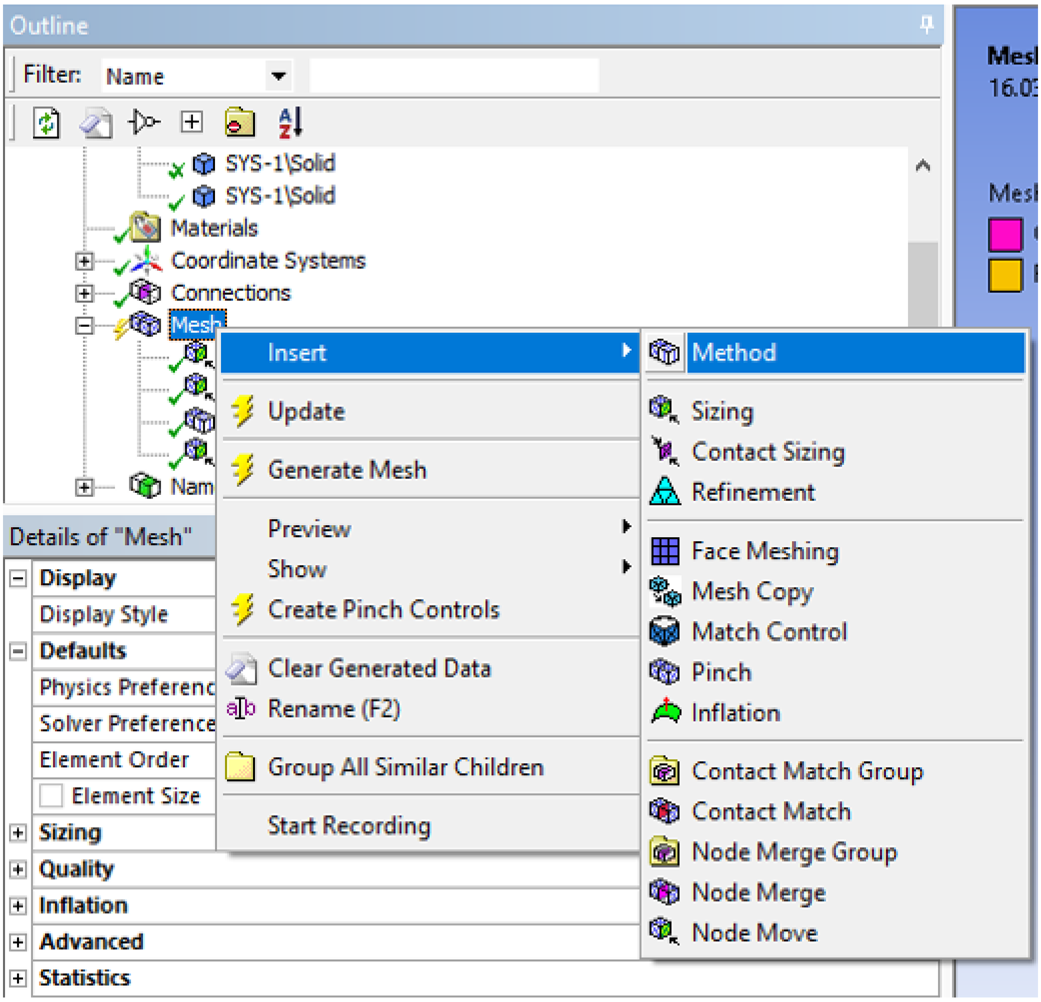
4) With LMB select the edge of the cuboid along the pipe direction, then confirm *Apply*



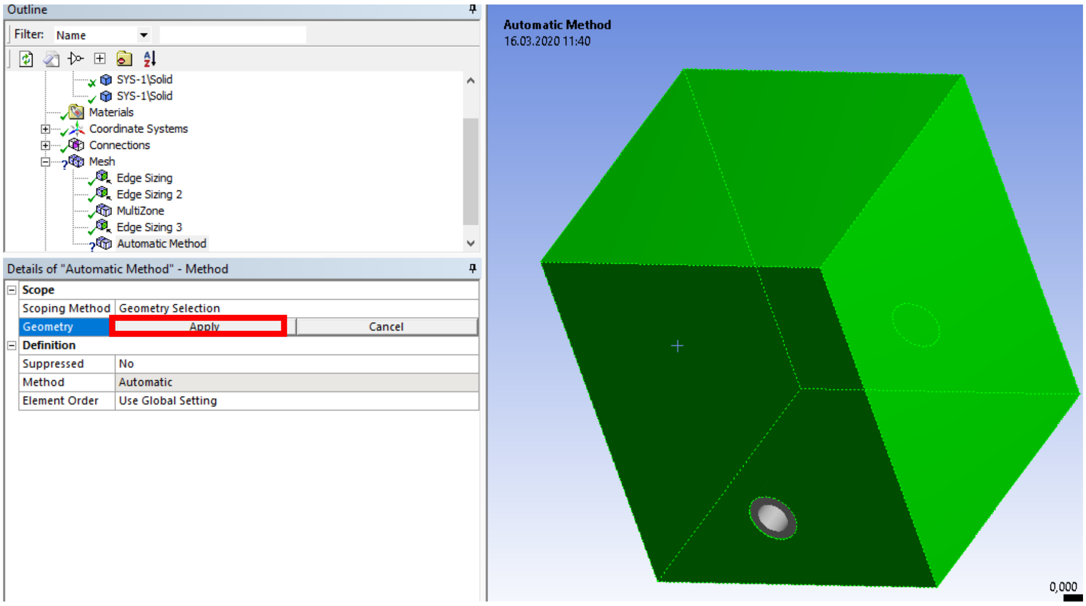
5) In *Details* apply the following settings



6) Click *Generate Mesh* and observe the numerical mesh.
7) Click RMB on *Mesh* and select *Insert->Method*



8) LMB select cuboid and confirm *Apply*.

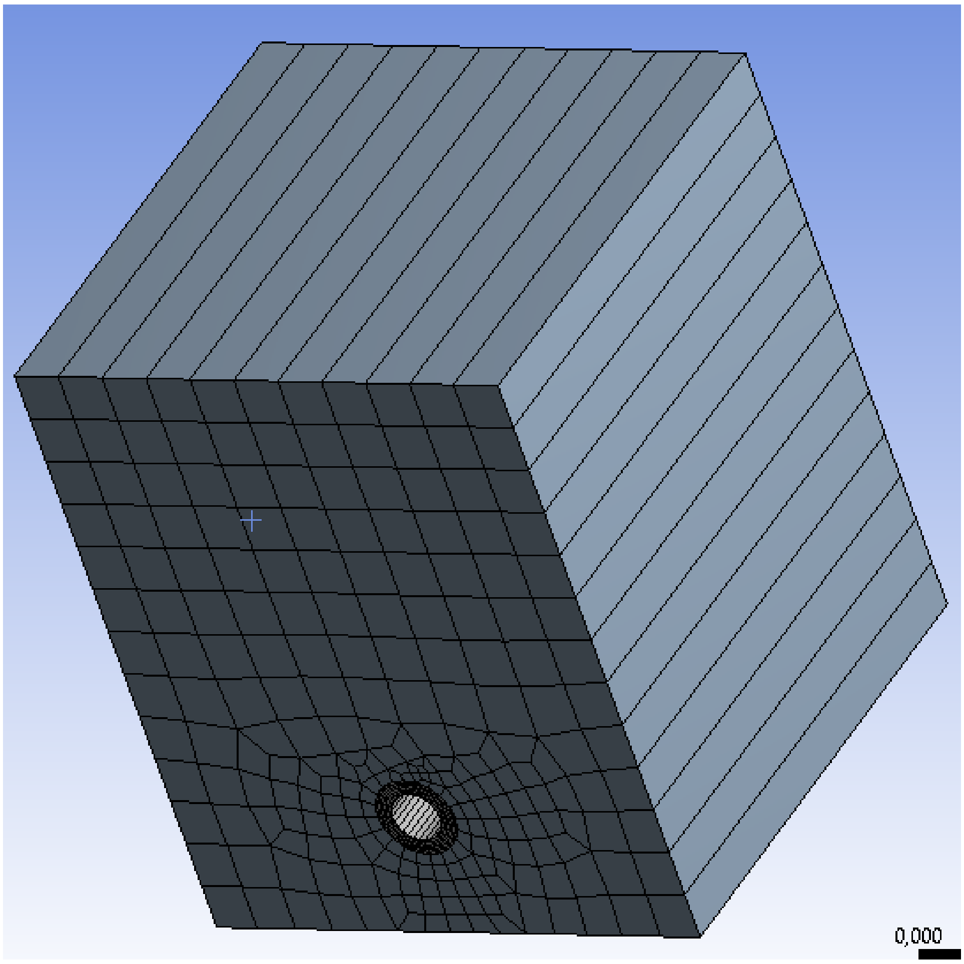


9) In *Details* apply the settings below

Details of "MultiZone 2" - Method

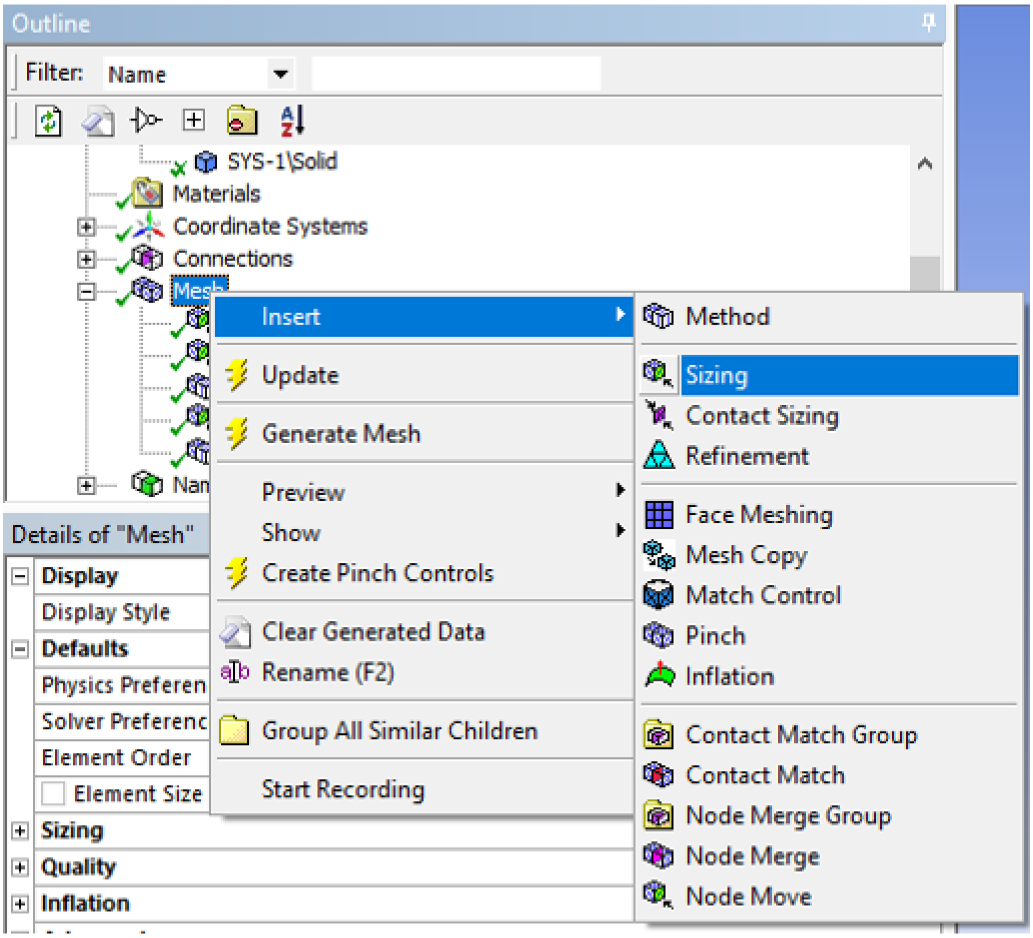
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Order	Use Global Setting
Src/Trg Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
Sweep Element Size	Default
Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off
Minimum Edge Length	5,0265e-002 m
Write ICEM CFD Files	No

10) Click *Generate Mesh* and observe the numerical mesh.



11) Change filter into body selection

12) RMB on *Mesh* and select *Insert->Sizing*

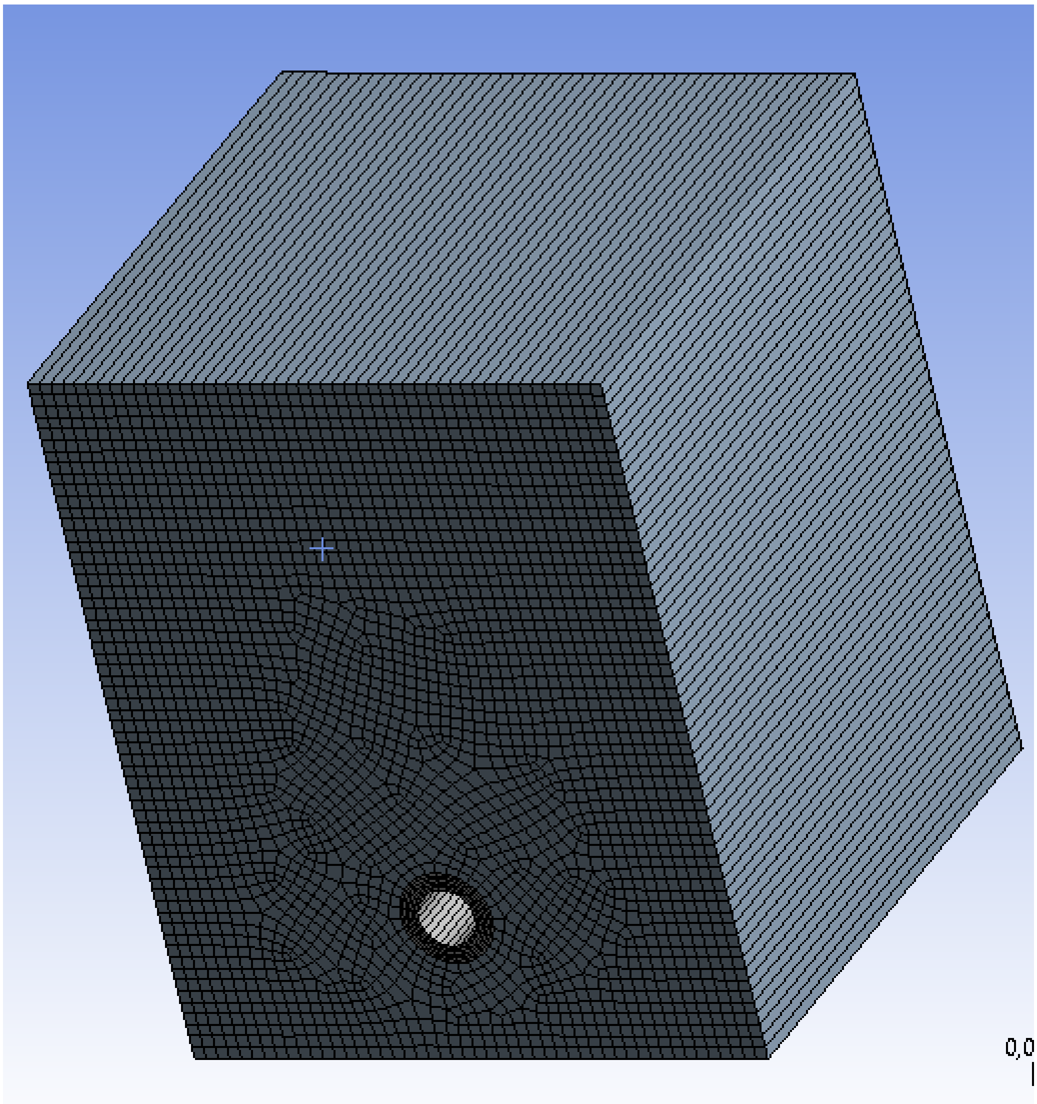


13) Select cuboid and confirm *Apply*.

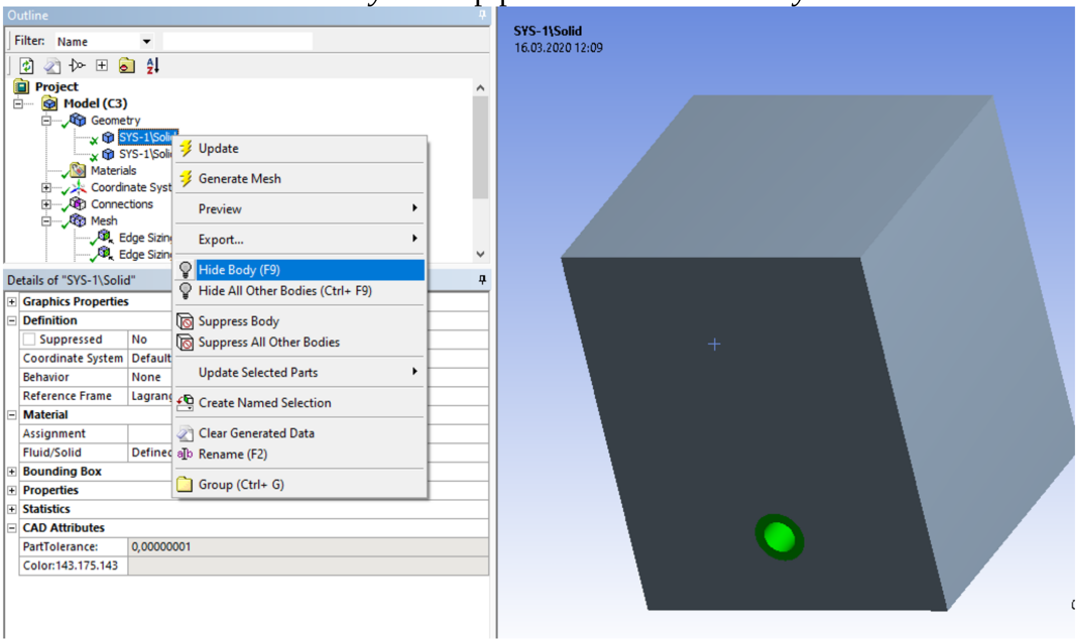
14) Apply the settings below.

Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	2,e-003 m
Advanced	
Defeature Size	Default (4,6368e-005 m)
Behavior	Hard
Capture Curvature	No
Capture Proximity	No

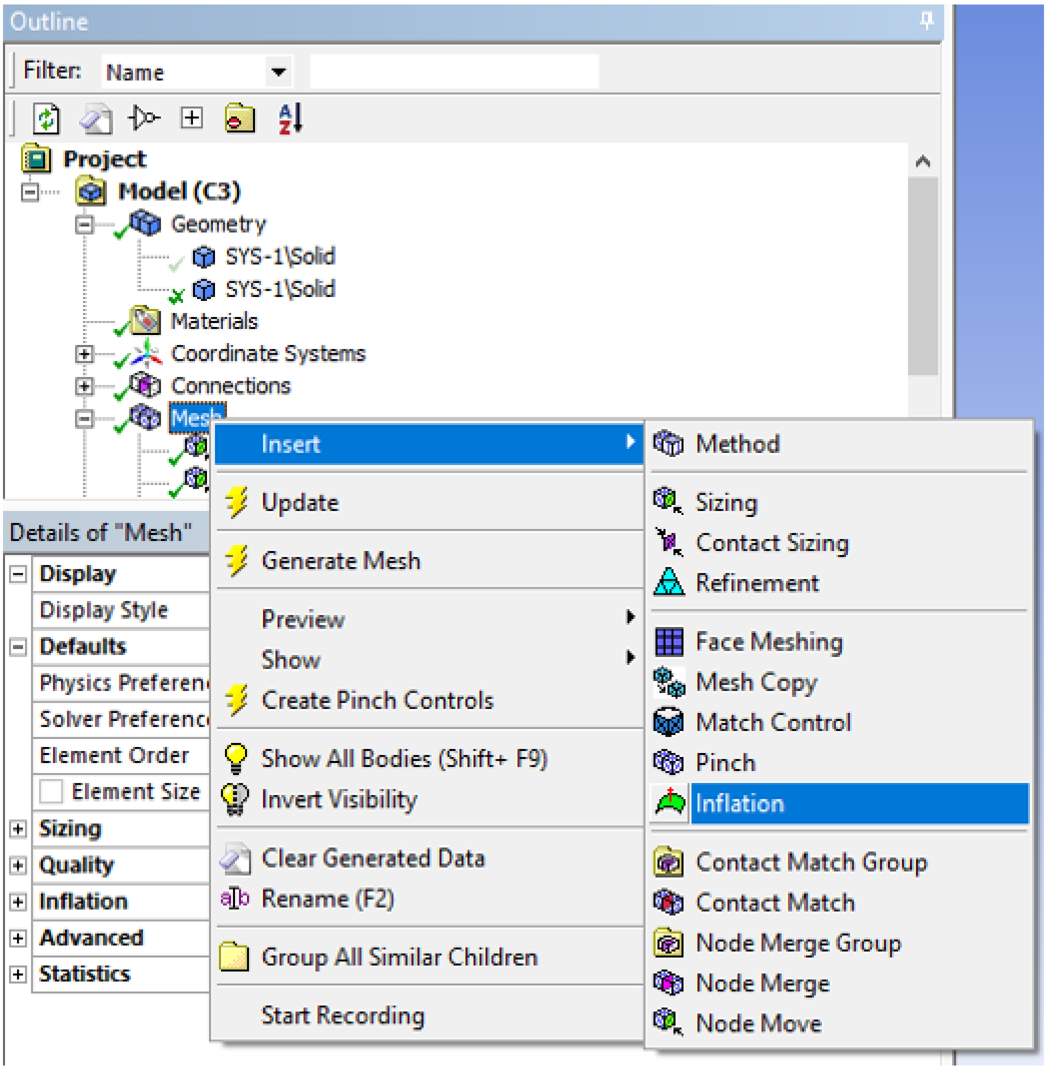
15) Click *Generat Mesh* and observe the numerical mesh.



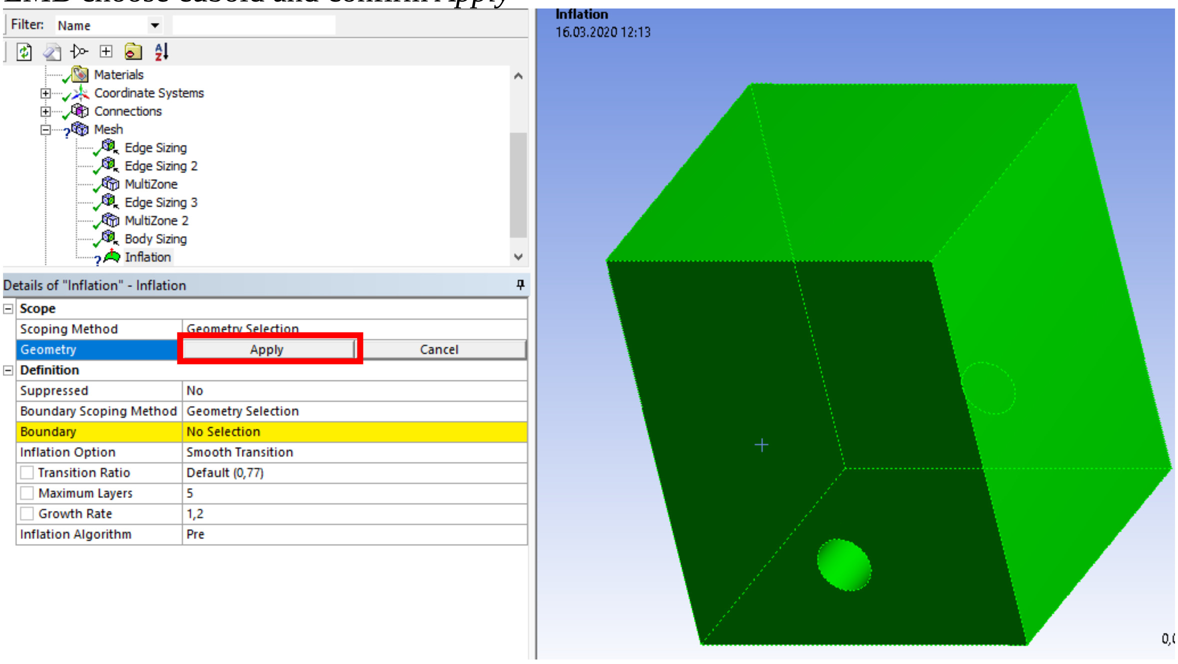
16) RMB in the tree on *Geometry* of the pipe and select *Hide Body*



17) RMB click on *Mesh* and select *Insert->Inflation*

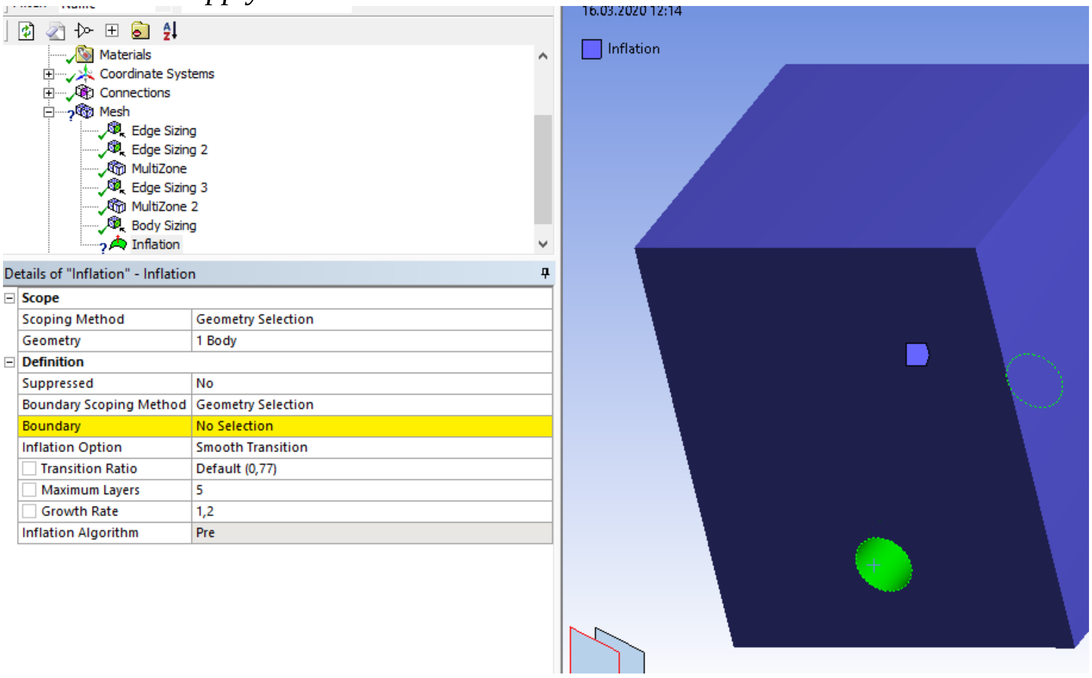


18) LMB choose cuboid and confirm *Apply*

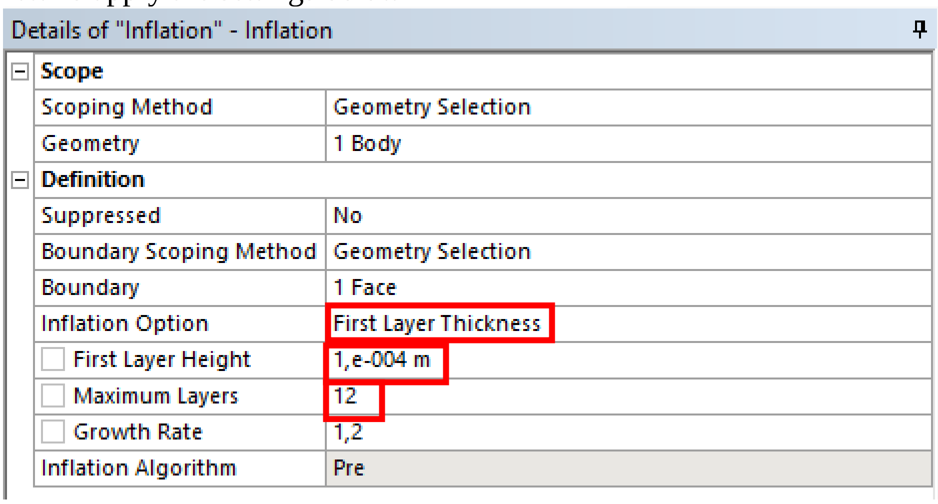


19) Change filter into face selection

20) LMB indicate the cylindrical surface in the cuboid and LMB click in the yellow field and then *Apply*



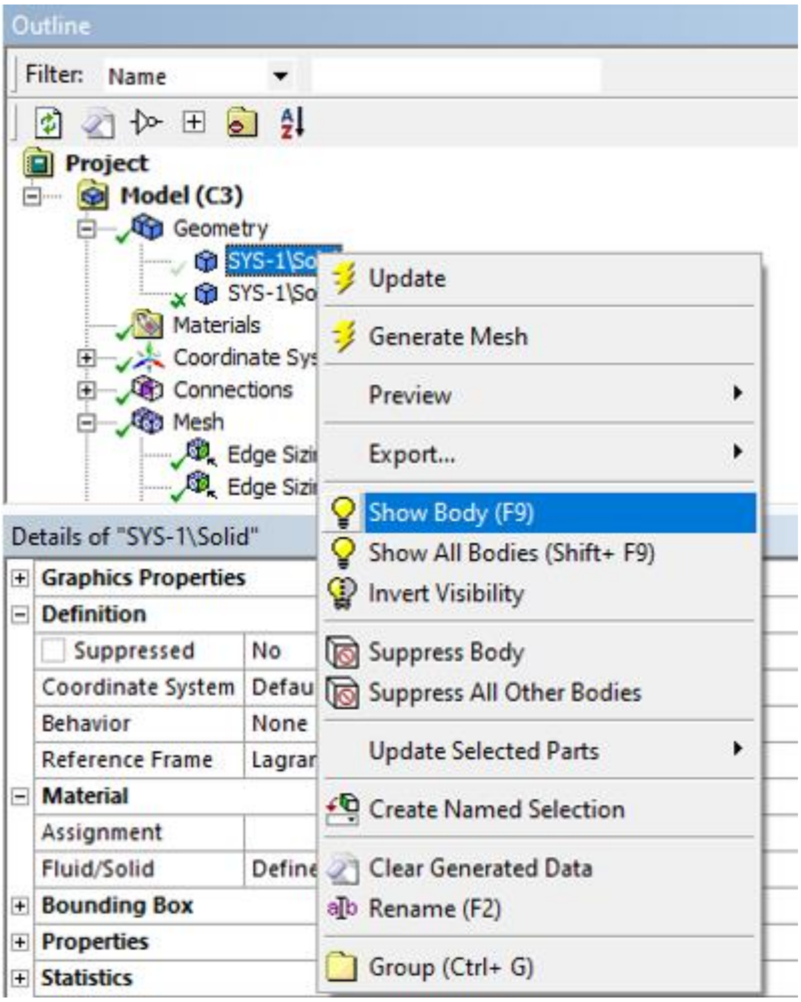
21) In *Details* apply the settings below



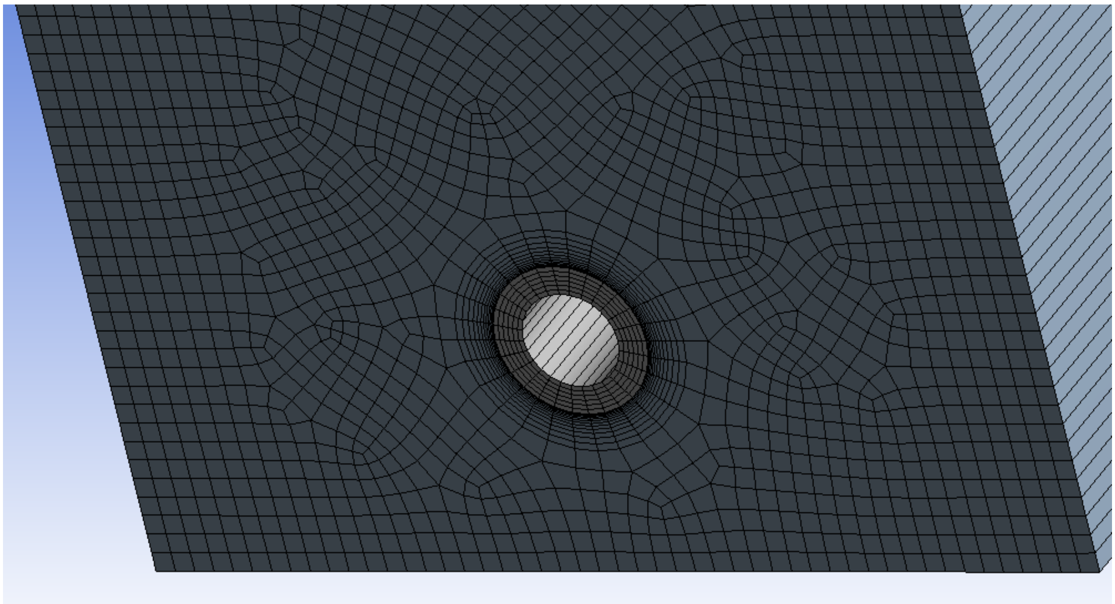
Details of "Inflation" - Inflation

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	1 Face
Inflation Option	First Layer Thickness
First Layer Height	1,e-004 m
Maximum Layers	12
Growth Rate	1,2
Inflation Algorithm	Pre

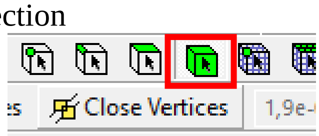
22) Turn on the visibility of the pipe



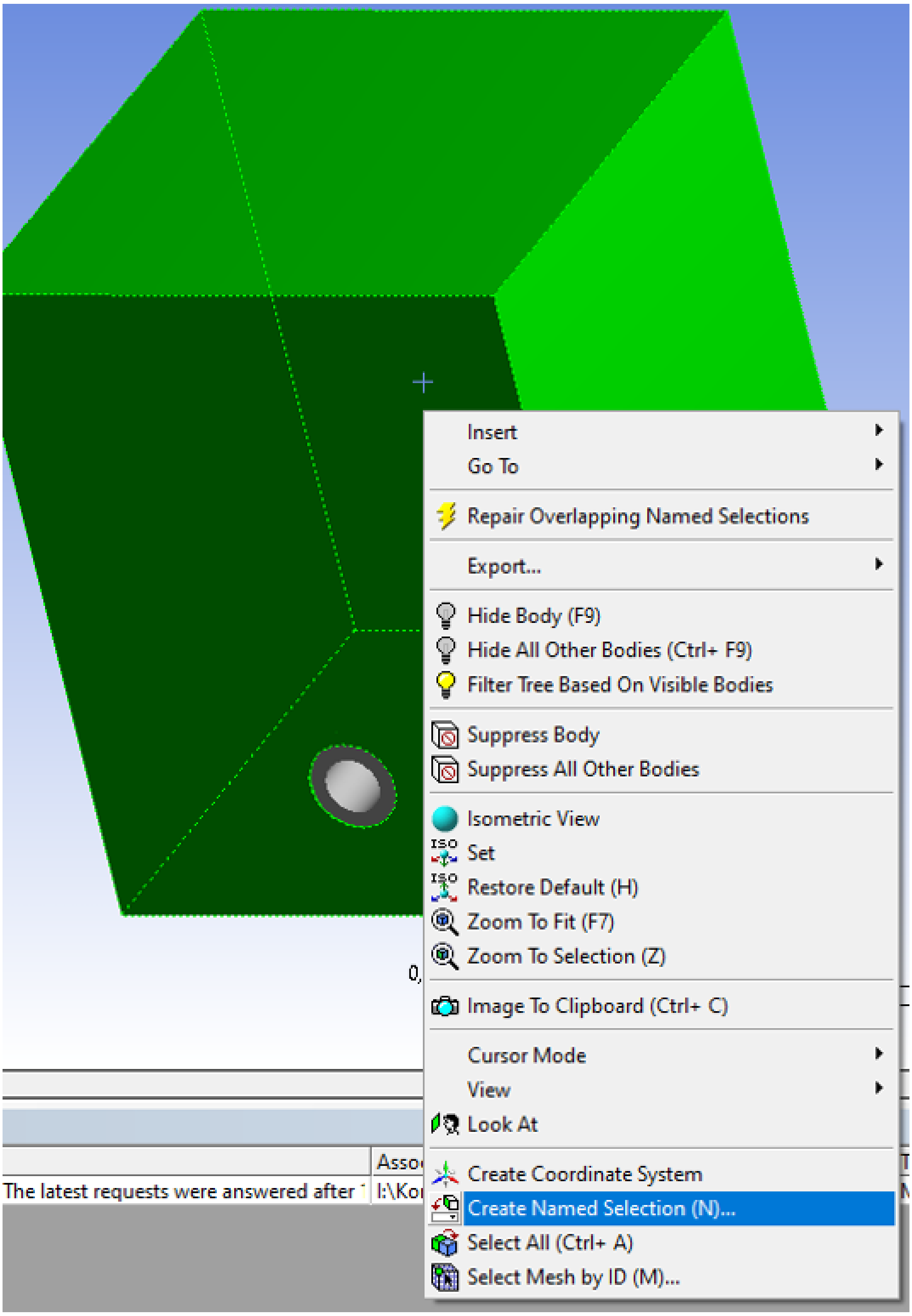
23) Click *Generate Mesh* and observe the numerical mesh.



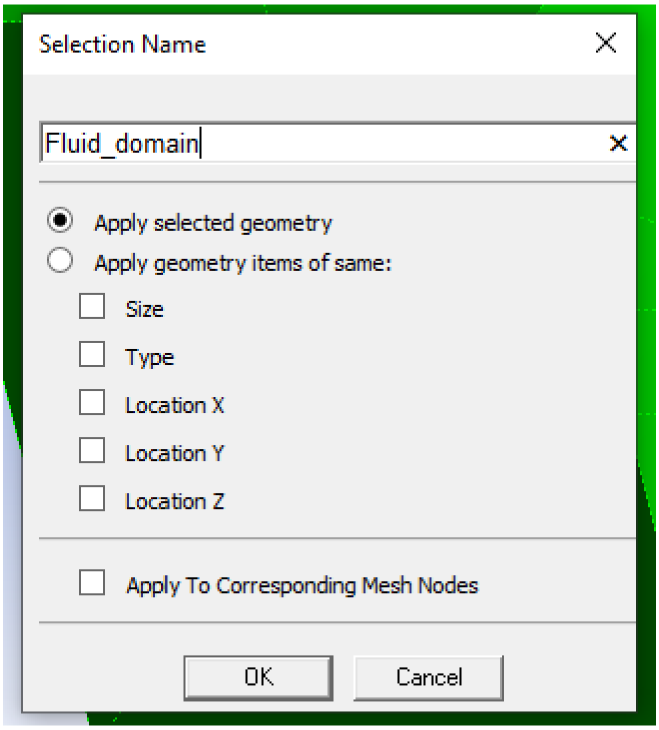
24) Change filter into body selection



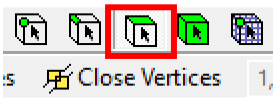
25) LMB choose cuboid and then RMB choose *Create Named Selection*



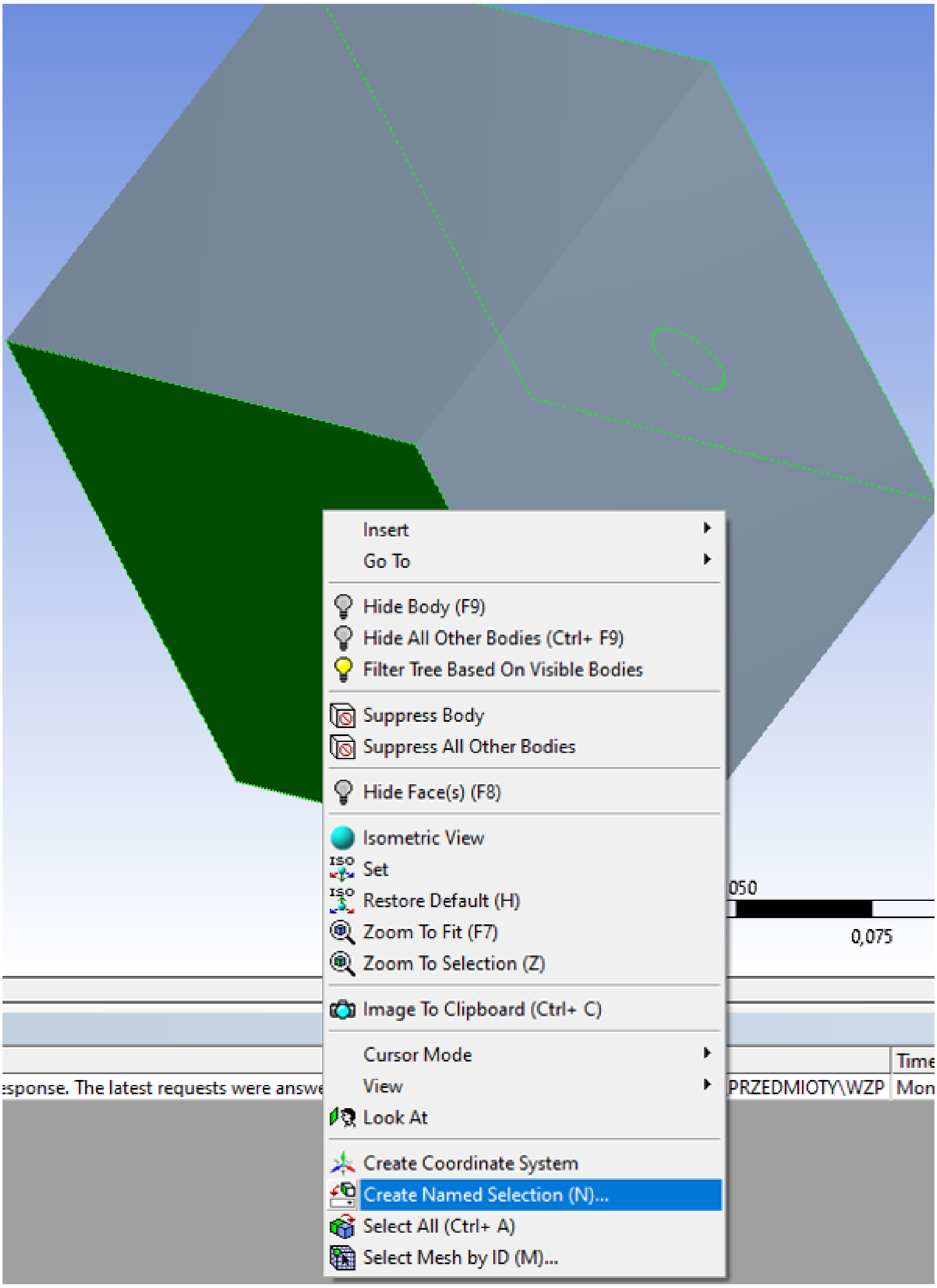
26) Name the body *Fluid_domain*



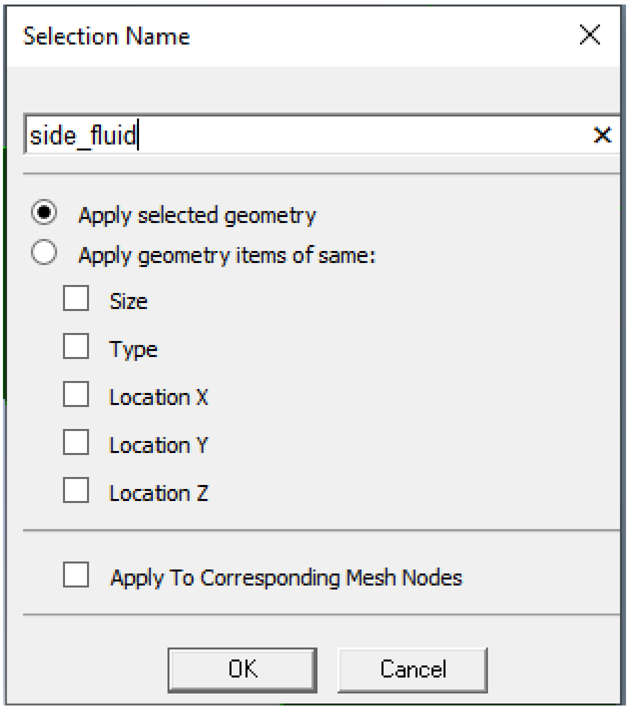
27) Change filter into face selection



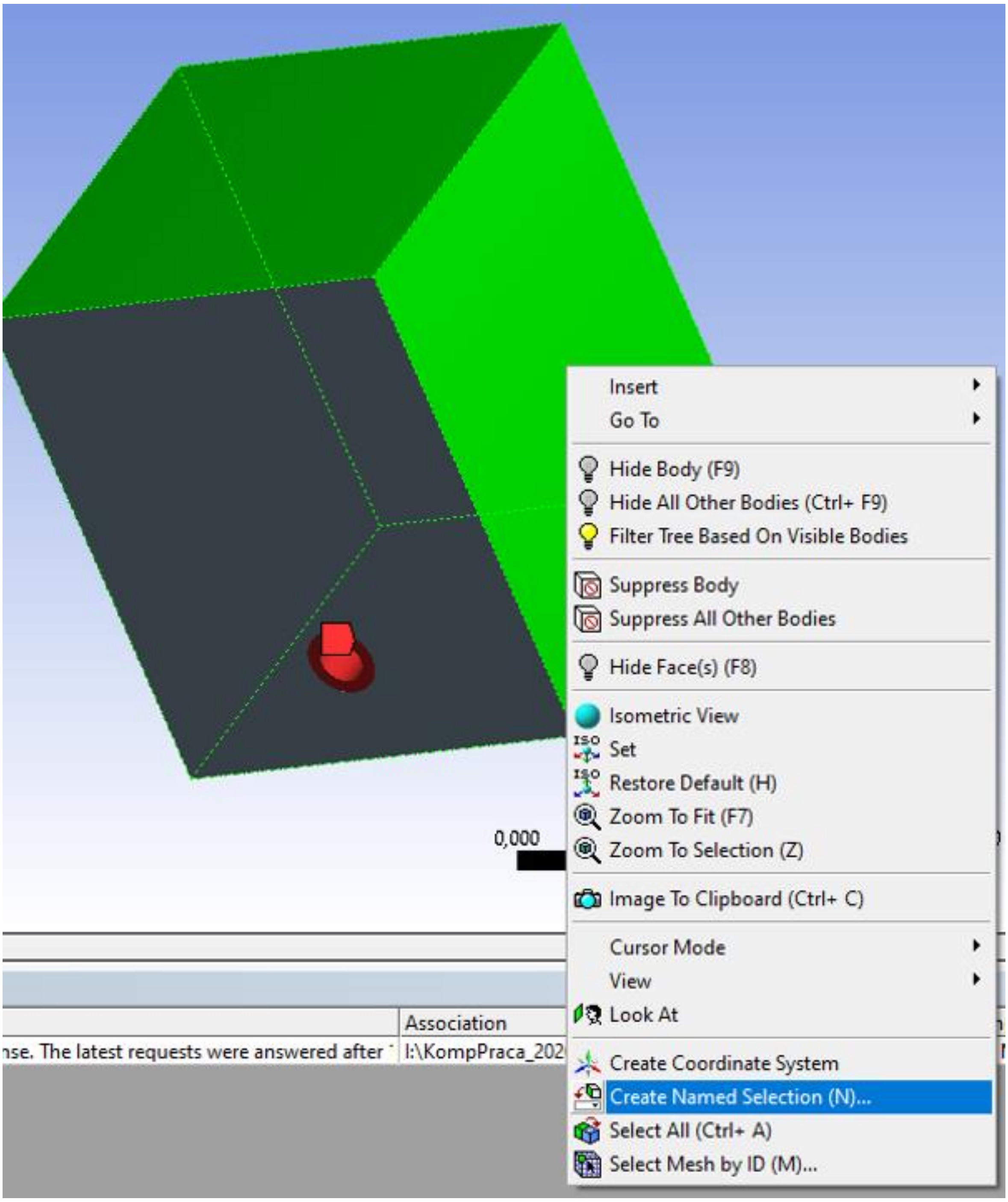
28) With *Ctrl* pressed choose two frontal faces in the cuboid



29) Name it *side_fluid*



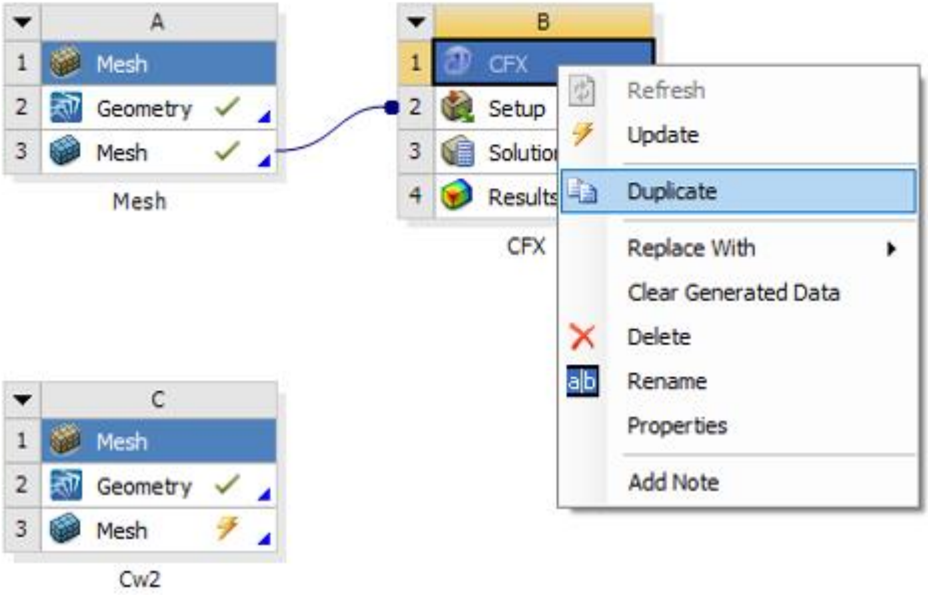
30) Similarly, select the remaining four walls of the cuboid and name them *walls*



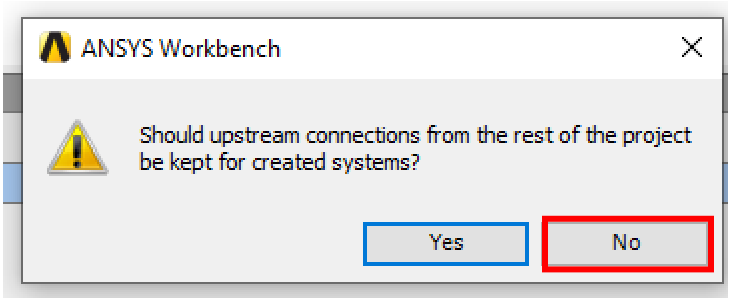
31) Close *Ansys Meshing* and save project in *Workbench.*

2.3. Symulacja numeryczna

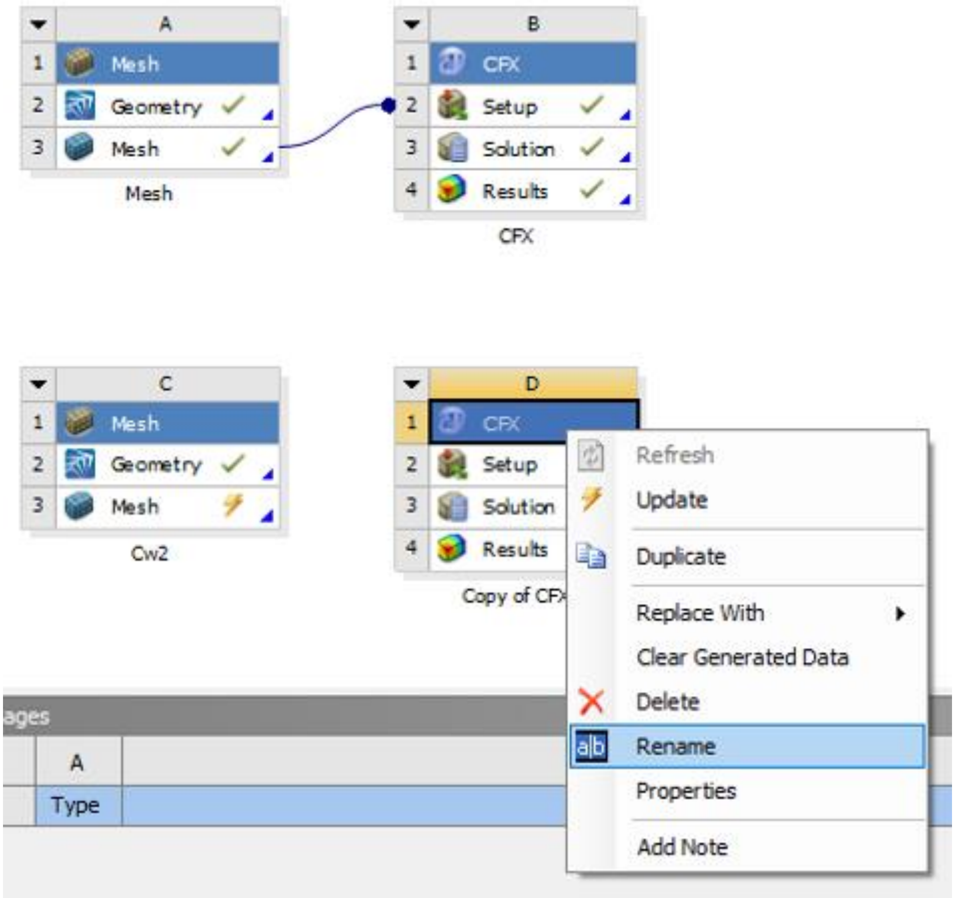
1) RMB on *CFX* from *Exercise no. 1* and select *Duplicate*



2) Then select *No*

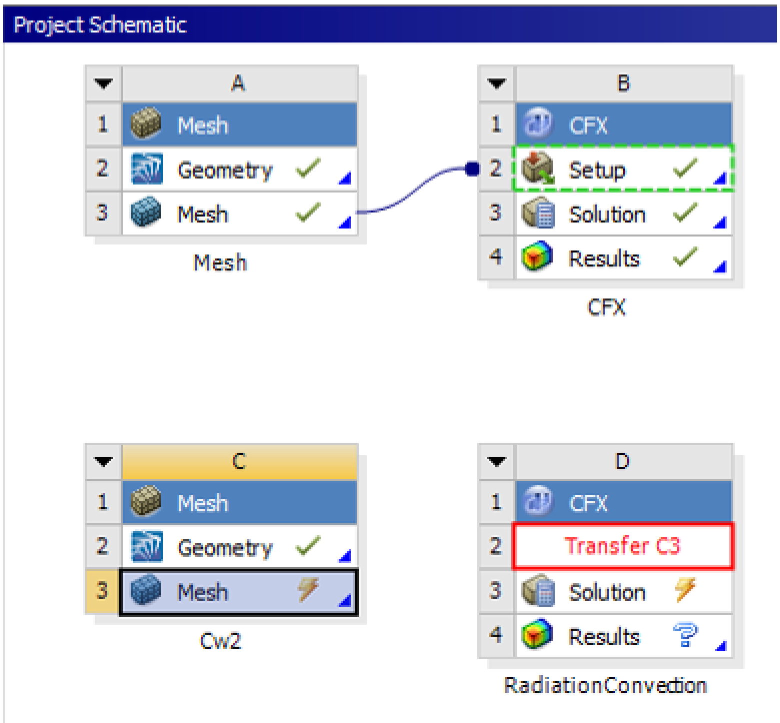


3) Change name into *RadiationConvection*

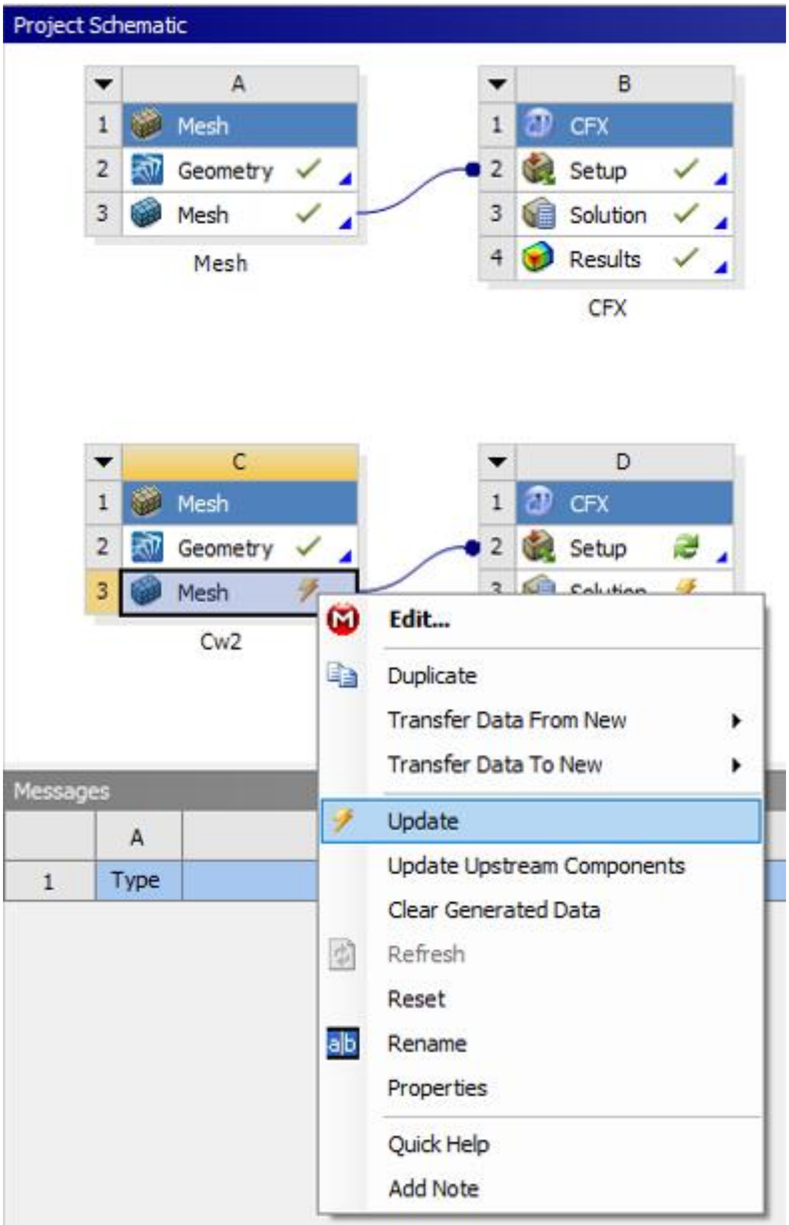


Note: if you don't see cell D, use Scroll to zoom out in *Project Schematic*. Then you can move cell D with LMB.

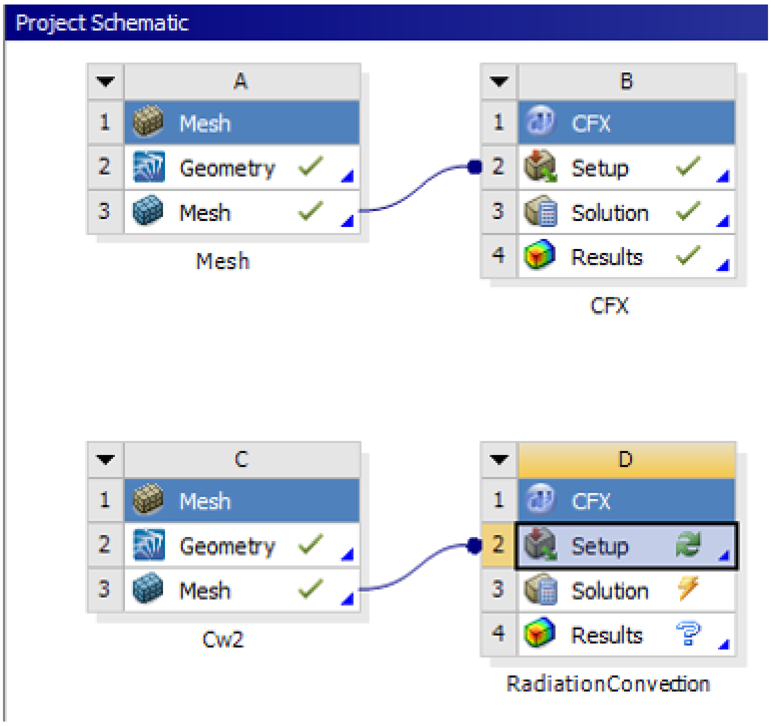
4) LMB click on *Mesh* in *Cw2* and drag on *Setup* till *Transfer C3* appears, next release LMB.



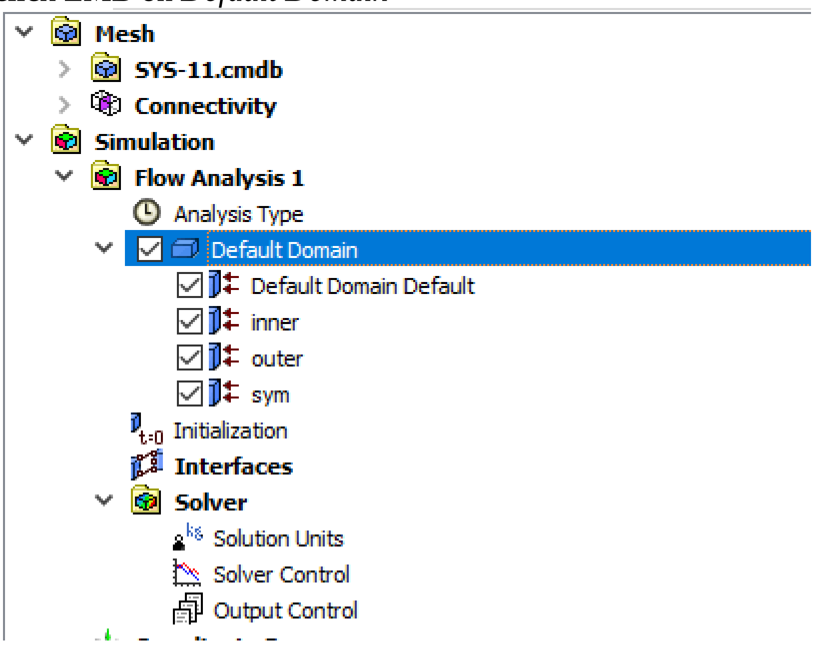
5) RMB on *Mesh* in *Cw2* and select *Update*



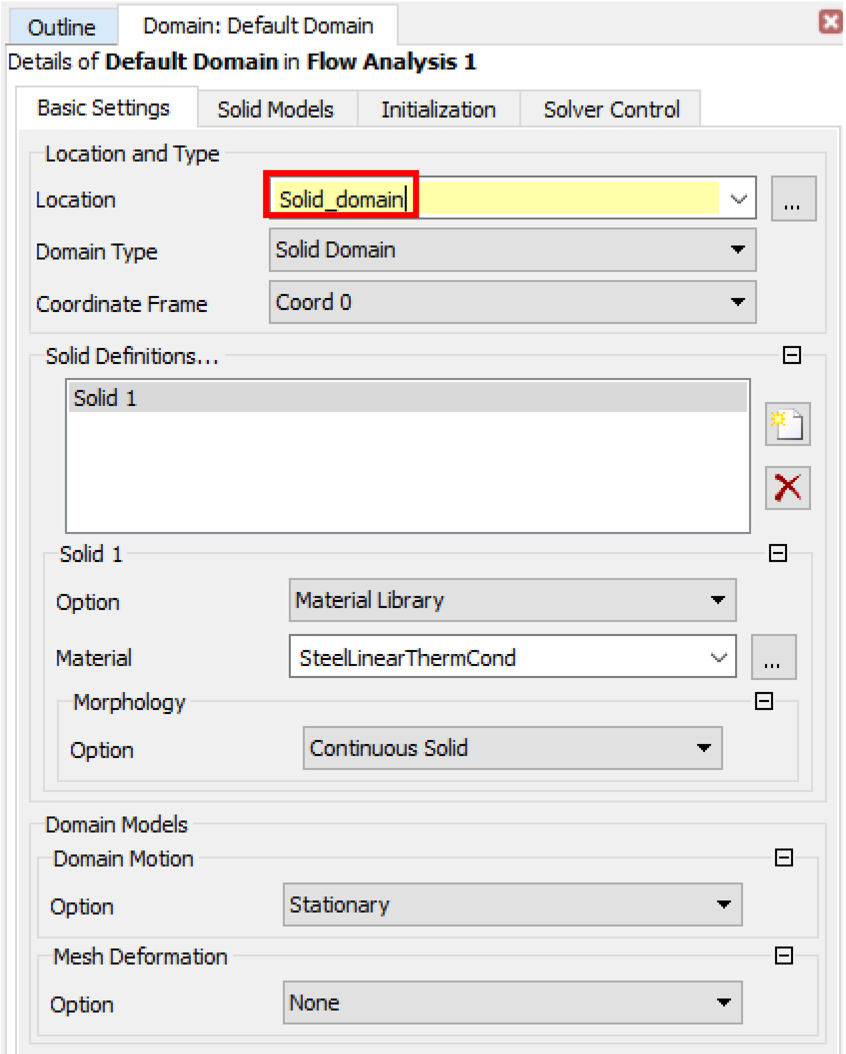
6) Double click LMB on *Setup* to edit a numerical model with a new numerical grid



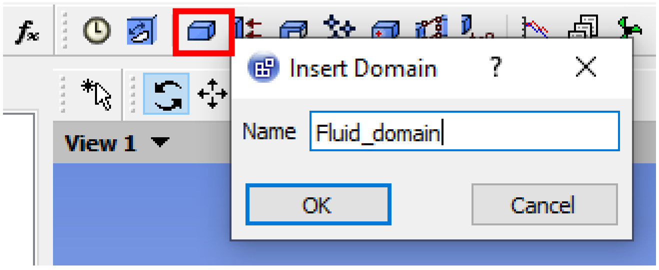
7) Double click LMB on *Default Domain*



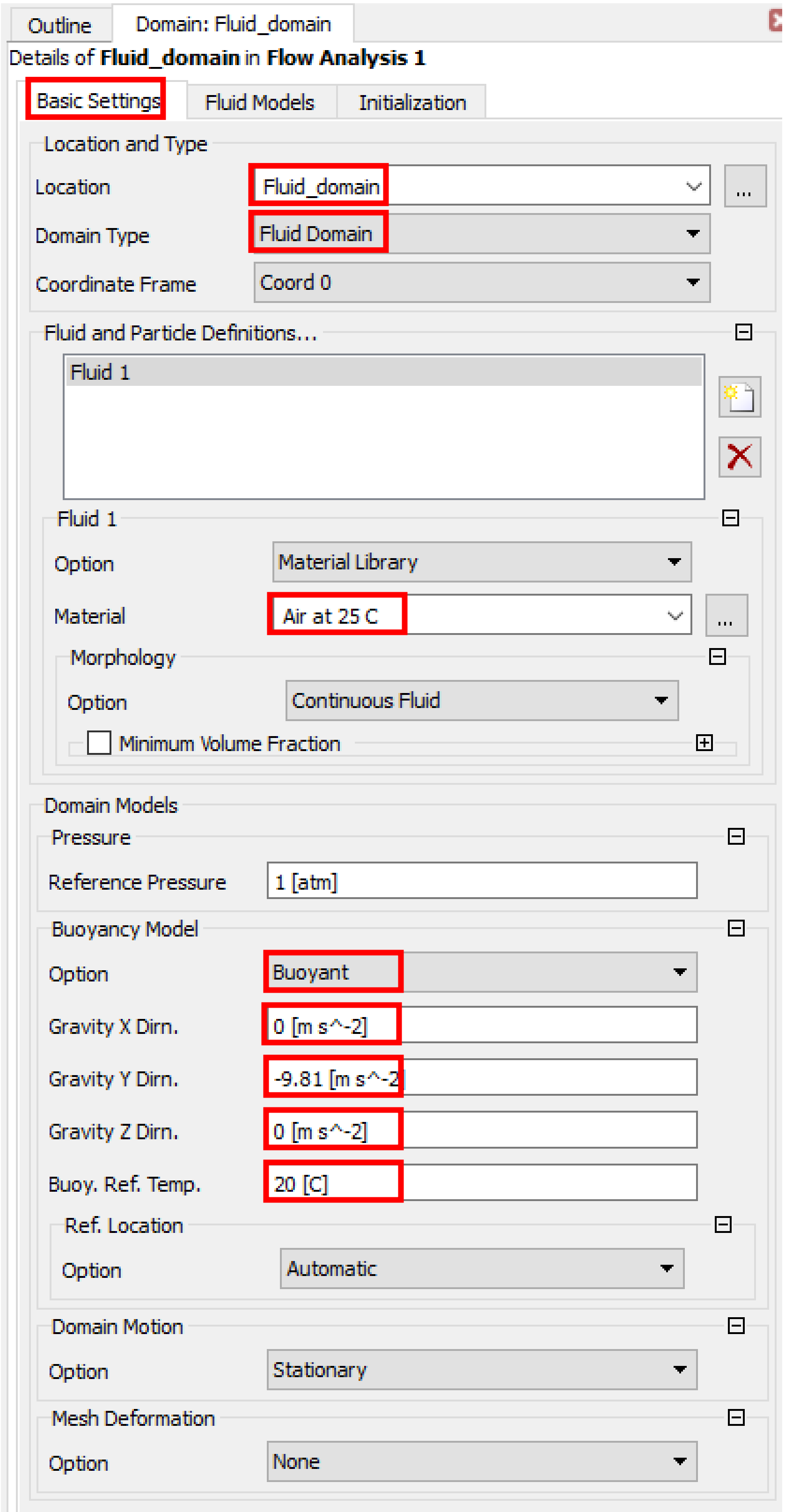
8) Apply the settings below.



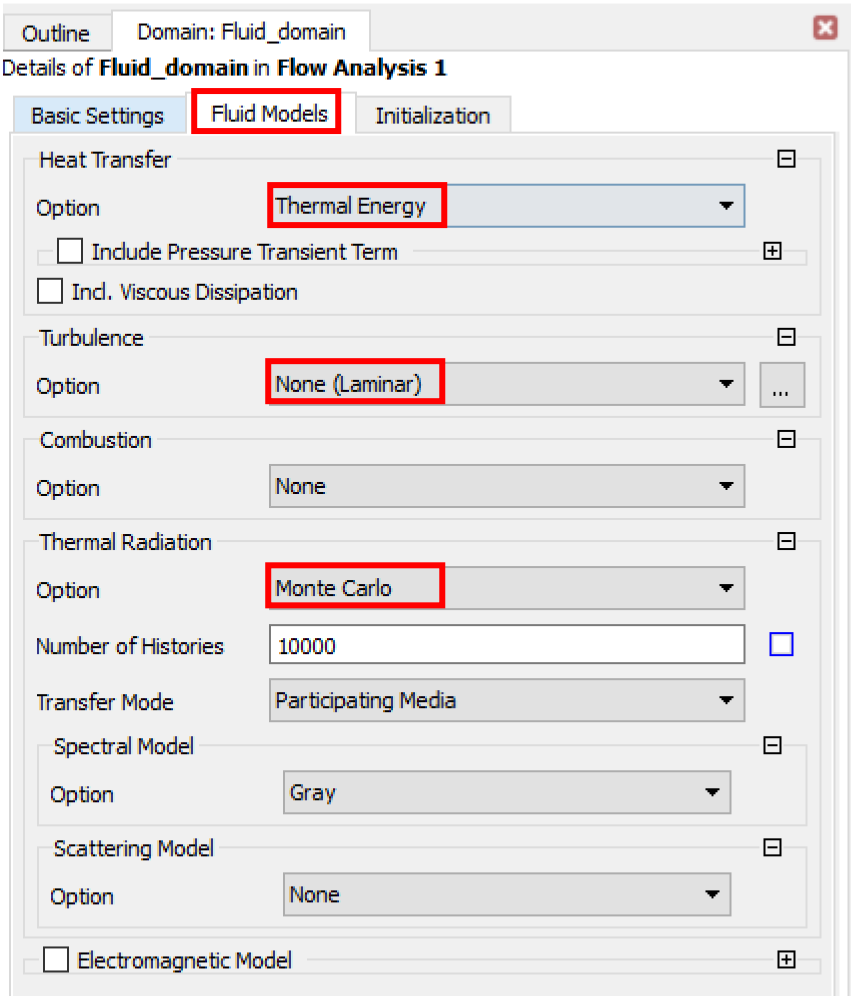
9) Create a fluid domain and name it *Fluid_domain*



10) Apply the settings below

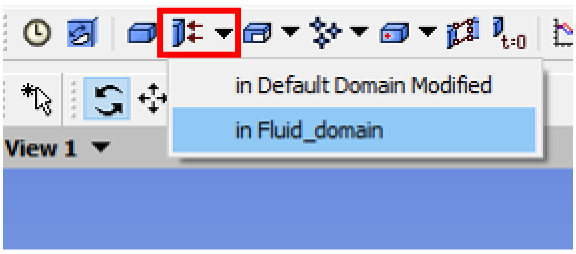


Outline
Domain: Fluid_domain
Details of Fluid_domain in Flow Analysis 1
Basic Settings
Fluid Models
Initialization
Location and Type
Location
Fluid_domain
Domain Type
Fluid Domain
Coordinate Frame
Coord 0
Fluid and Particle Definitions...
Fluid 1
Fluid 1
Option
Material Library
Material
Air at 25 C
Morphology
Option
Continuous Fluid
Minimum Volume Fraction
Domain Models
Pressure
Reference Pressure
1 [atm]
Buoyancy Model
Option
Buoyant
Gravity X Dirn.
0 [m s^-2]
Gravity Y Dirn.
-9.81 [m s^-2
Gravity Z Dirn.
0 [m s^-2]
Buoy. Ref. Temp.
20 [C]
Ref. Location
Option
Automatic
Domain Motion
Option
Stationary
Mesh Deformation
Option
None

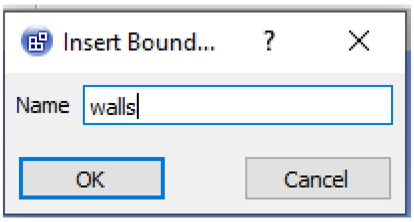


11) Confirm *OK*.

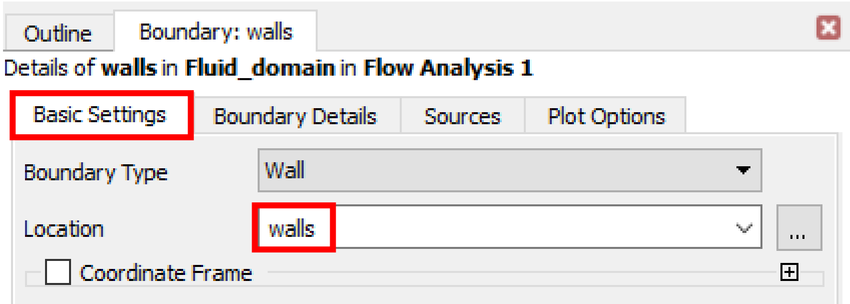
12) Create a boundary condition in the fluid domain

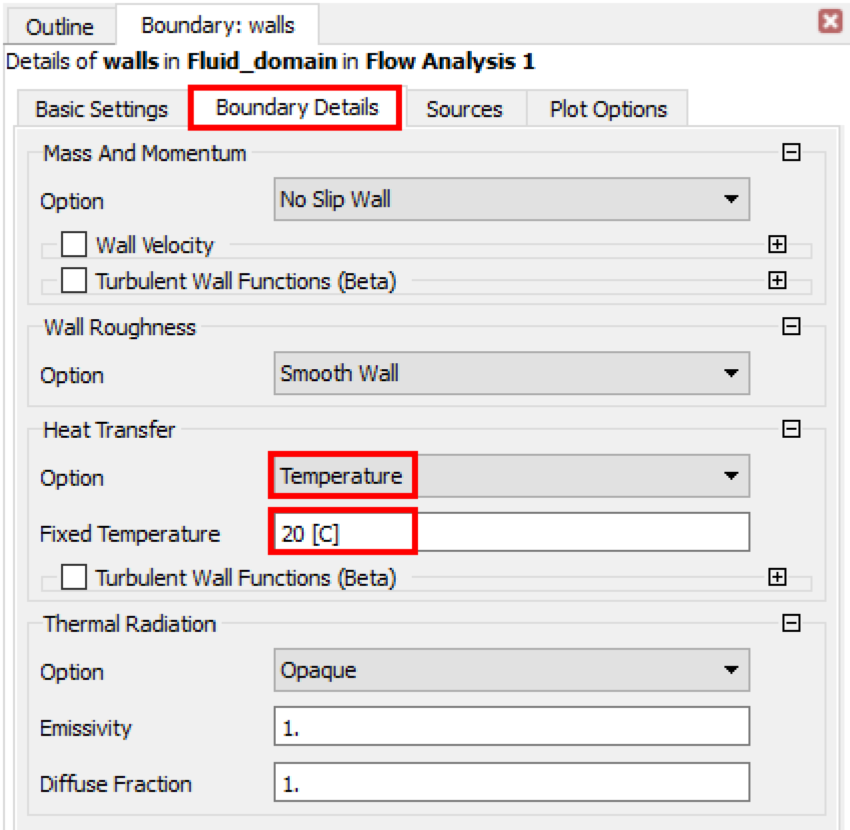


13) Name it *walls*

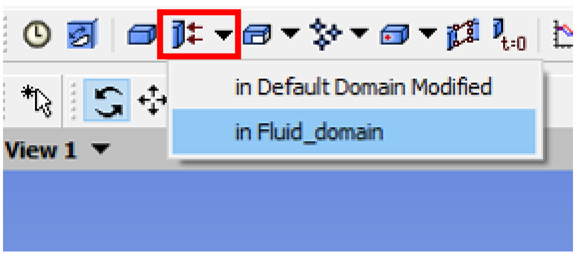


14) Make the following settings and confirm *OK*.

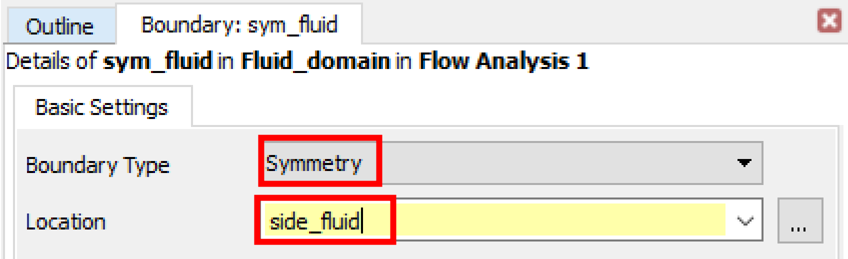




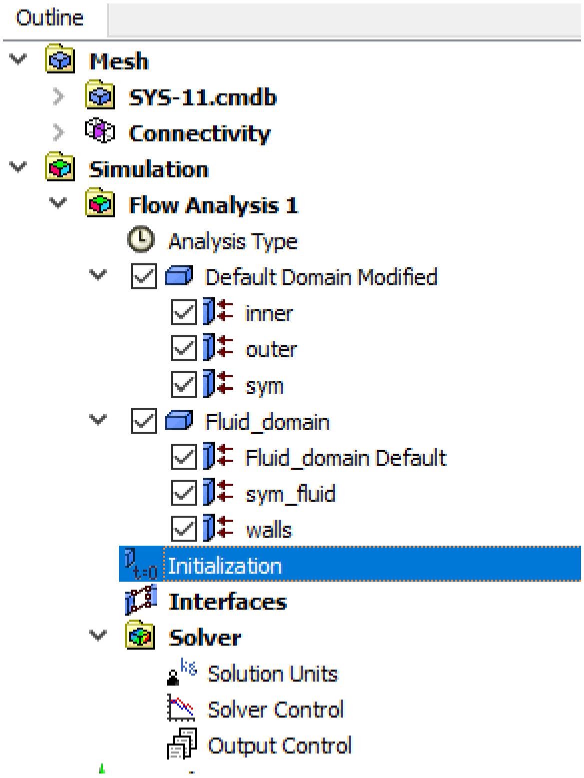
15) Create a boundary condition in the fluid domain



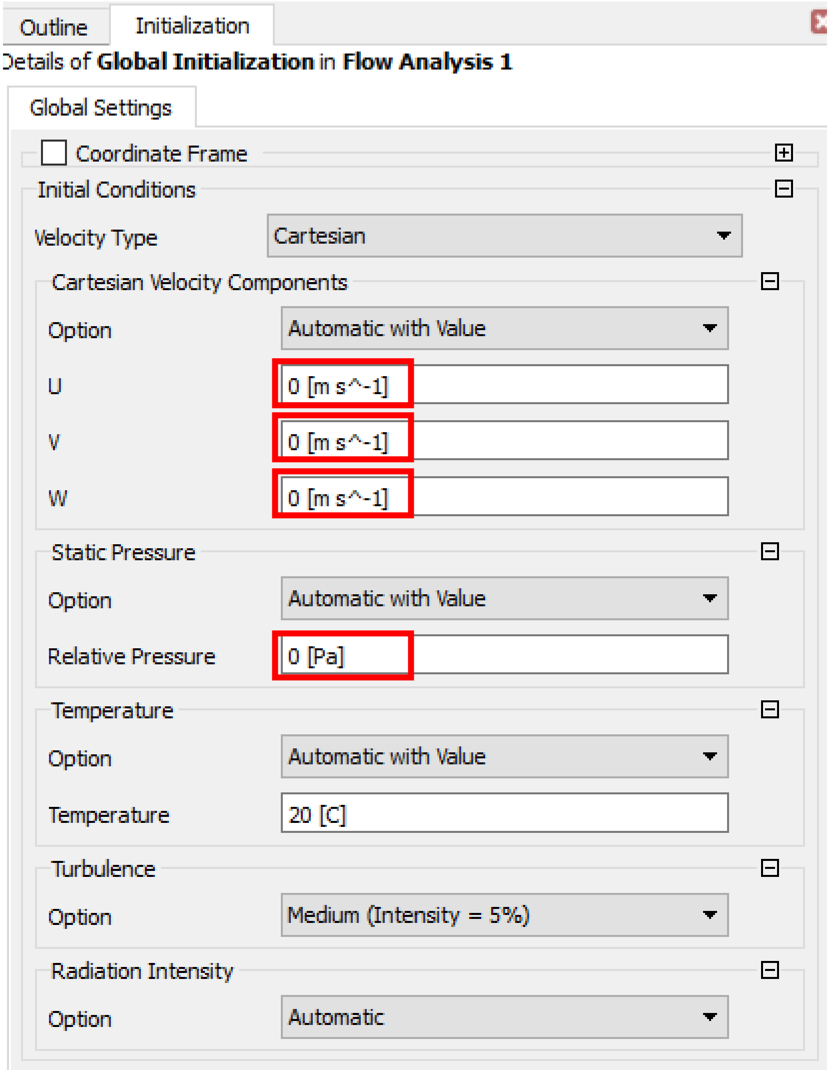
16) Name it *sym_fluid* and apply the settings below and confirm *OK*.



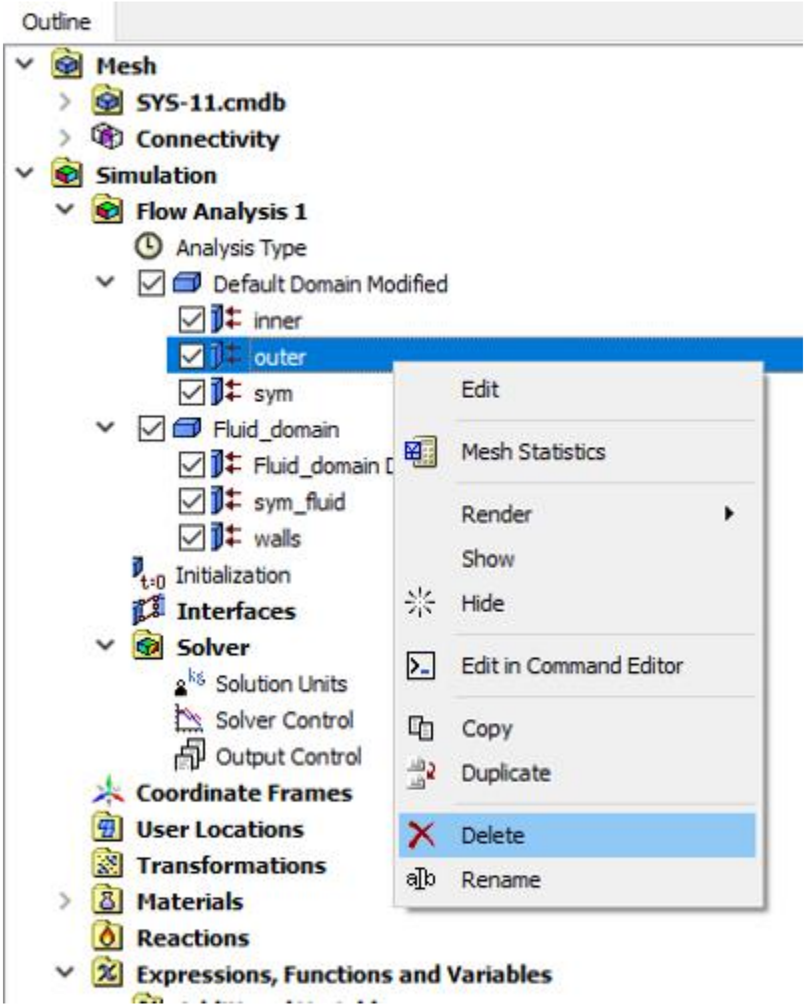
17) Double-click the initial condition icon



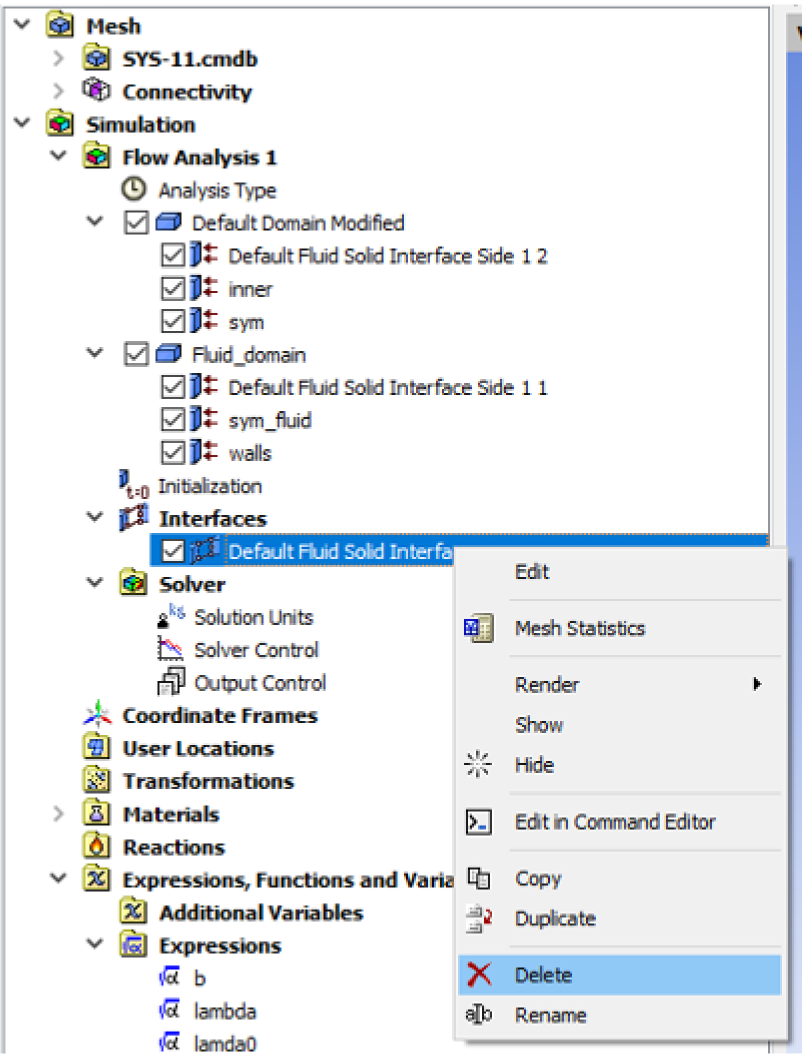
18) Apply the settings below and confirm *OK*.



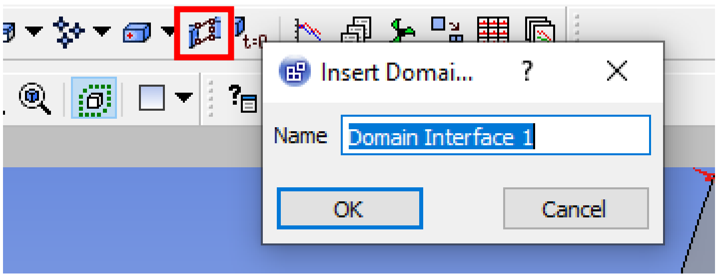
19) Delete *outer* boundary condition



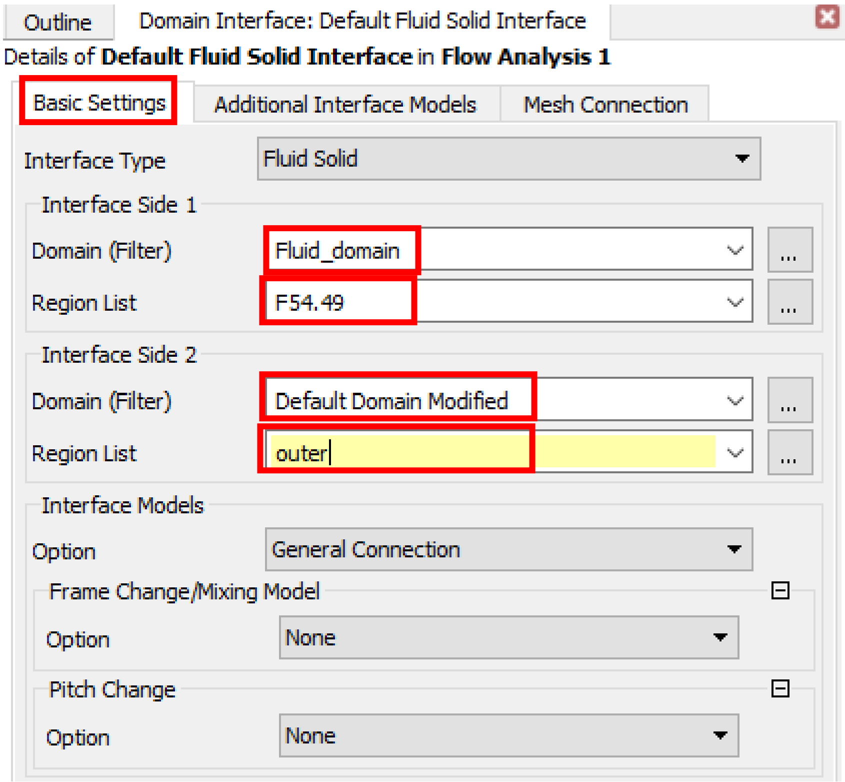
20) Delete the default settings for *Interface*

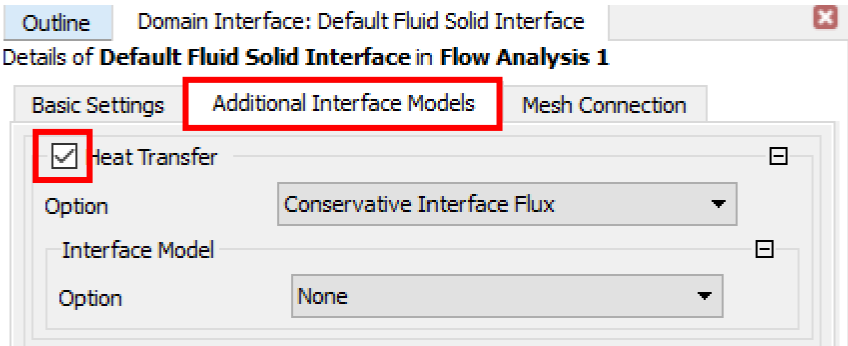


21) Create a new interface of fluid and solid area

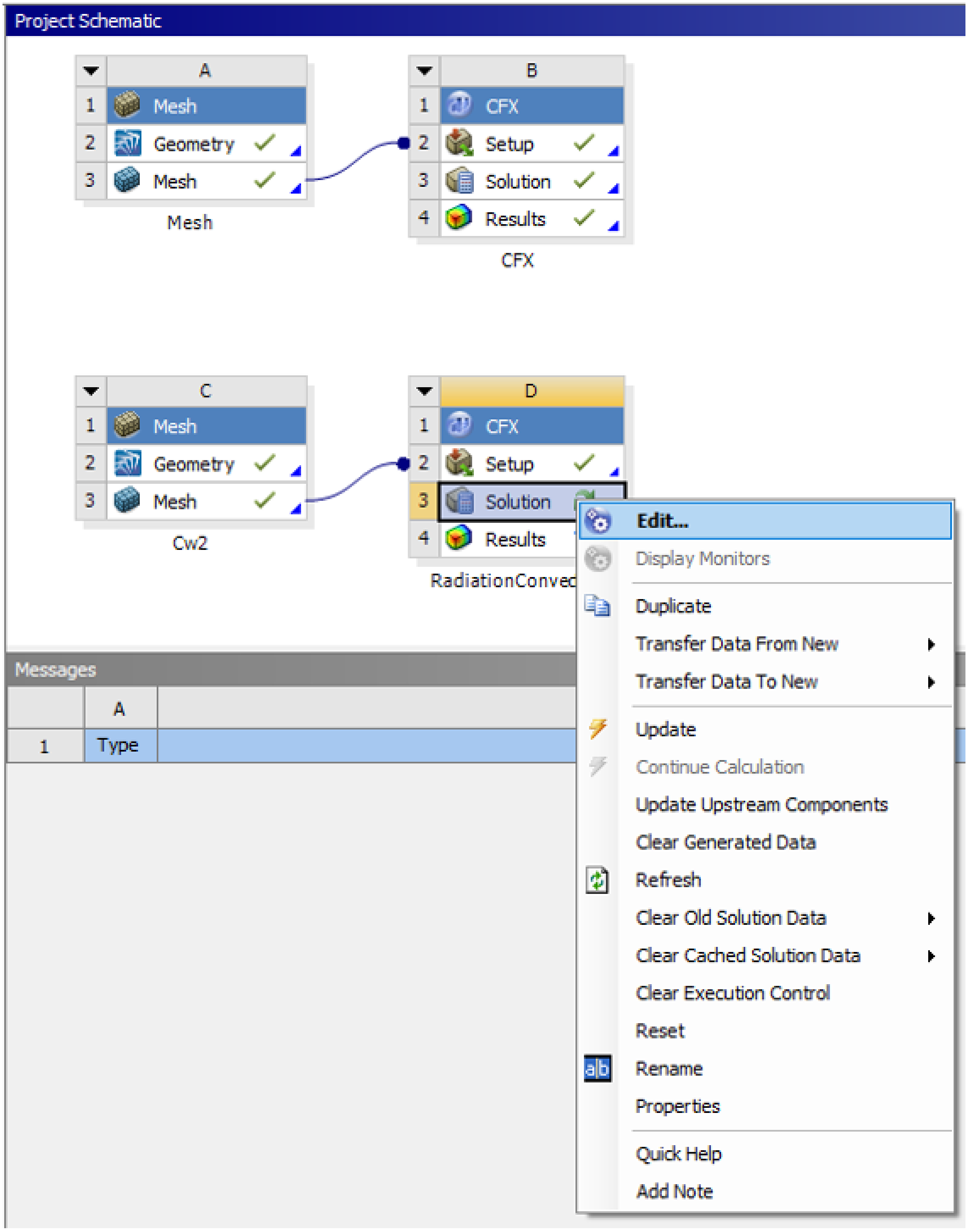


22) Apply the settings below

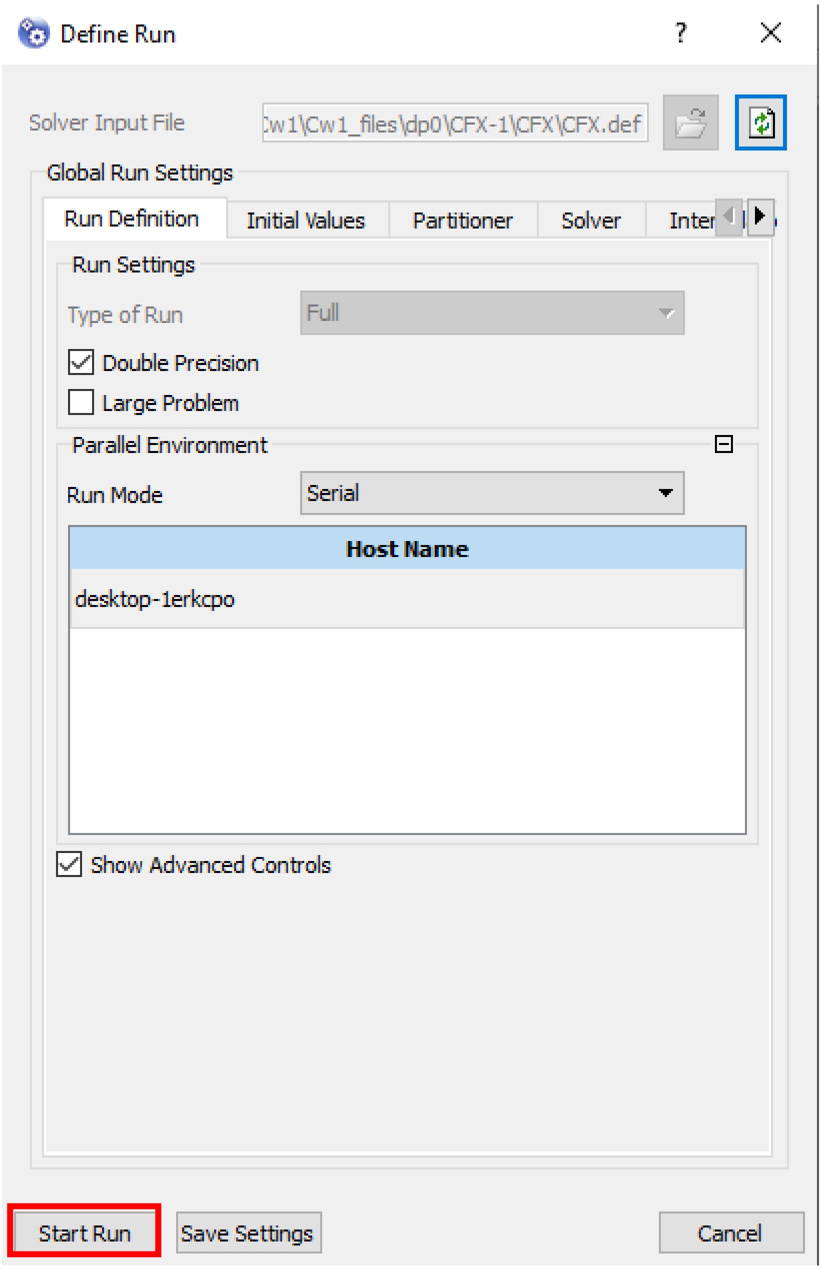




23) Confirm *OK*.
24) Close *Ansys CFX* and save project in *Workbench*.
25) With RMB edit *Solution*.



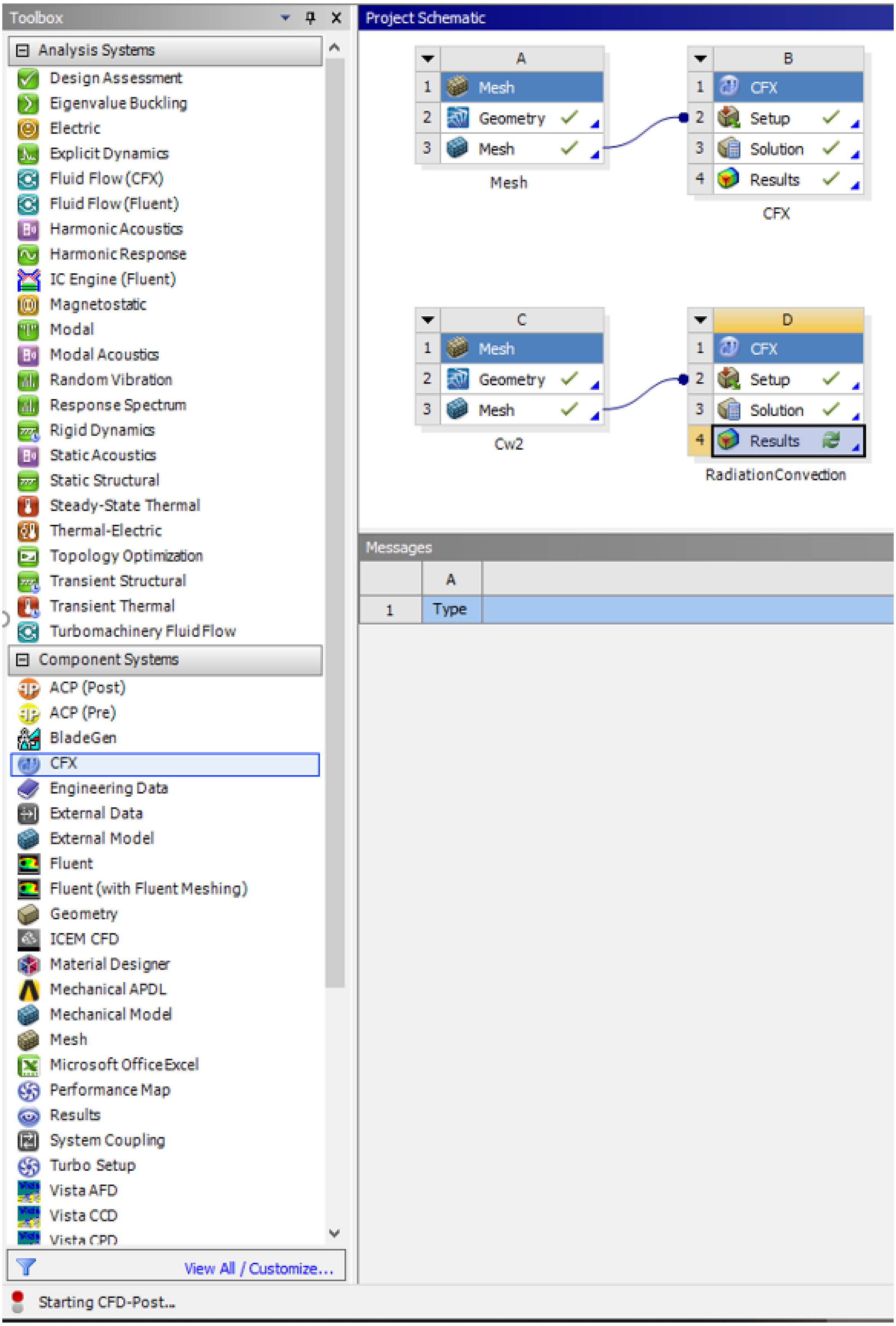
26) Press *Start run*



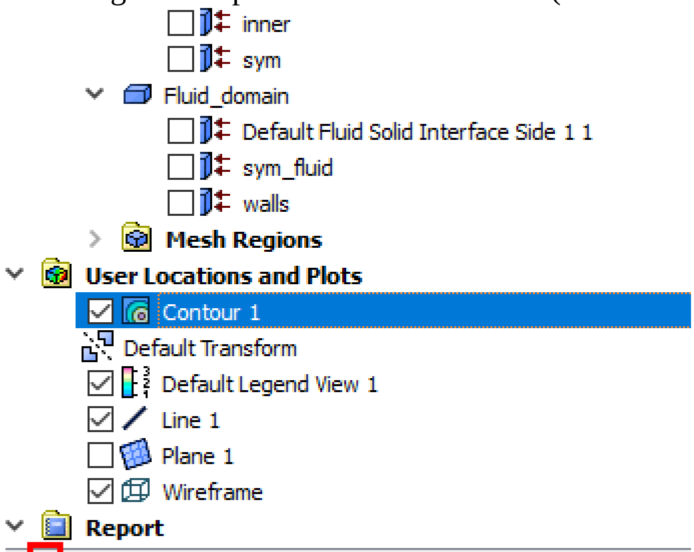
27) Wait for the program finish the calculations.
28) After completing the calculations, close the *Ansys CFX Solver Manager*.

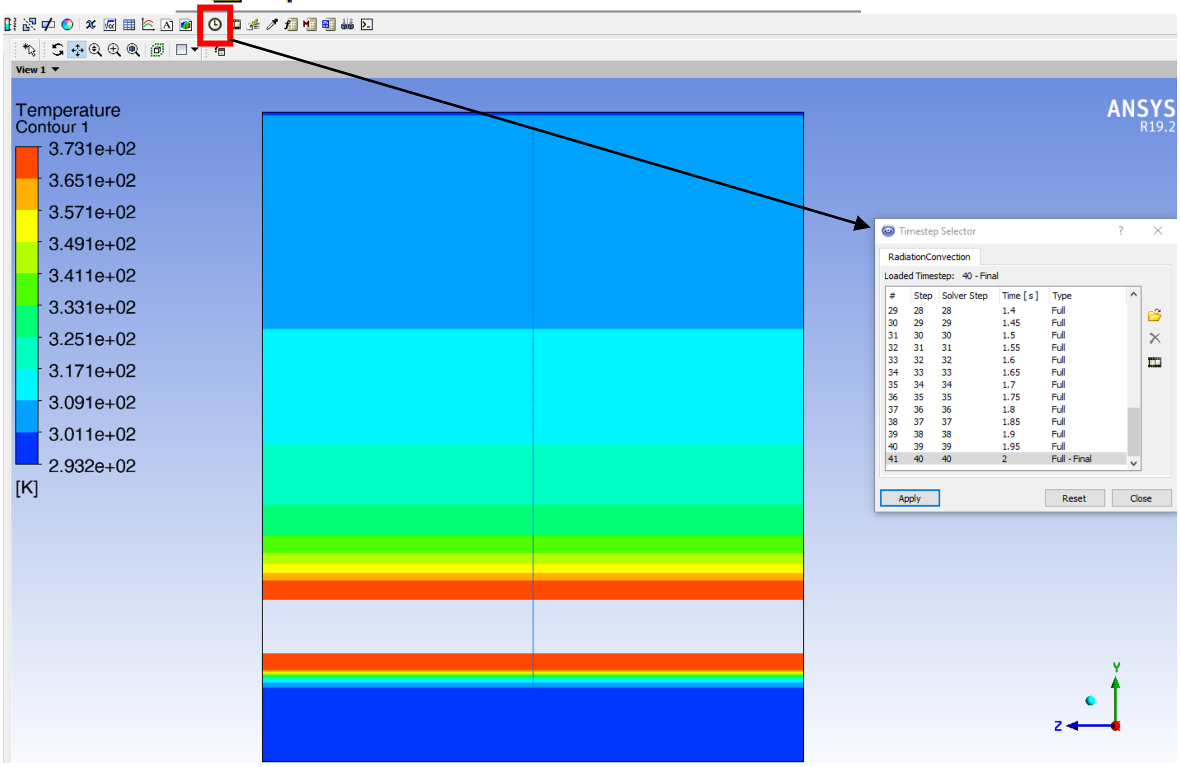
3. Results visualisation

1) Double click LMB on *Results*.

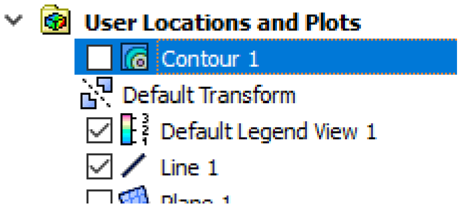


2) Click LMB the X axis in the right bottom corner of the screen.
3) Enable contours created in *Exercise no. 1* and observe the temperature distribution in the longitudinal plane for different times (icon *Timestep Selector*)

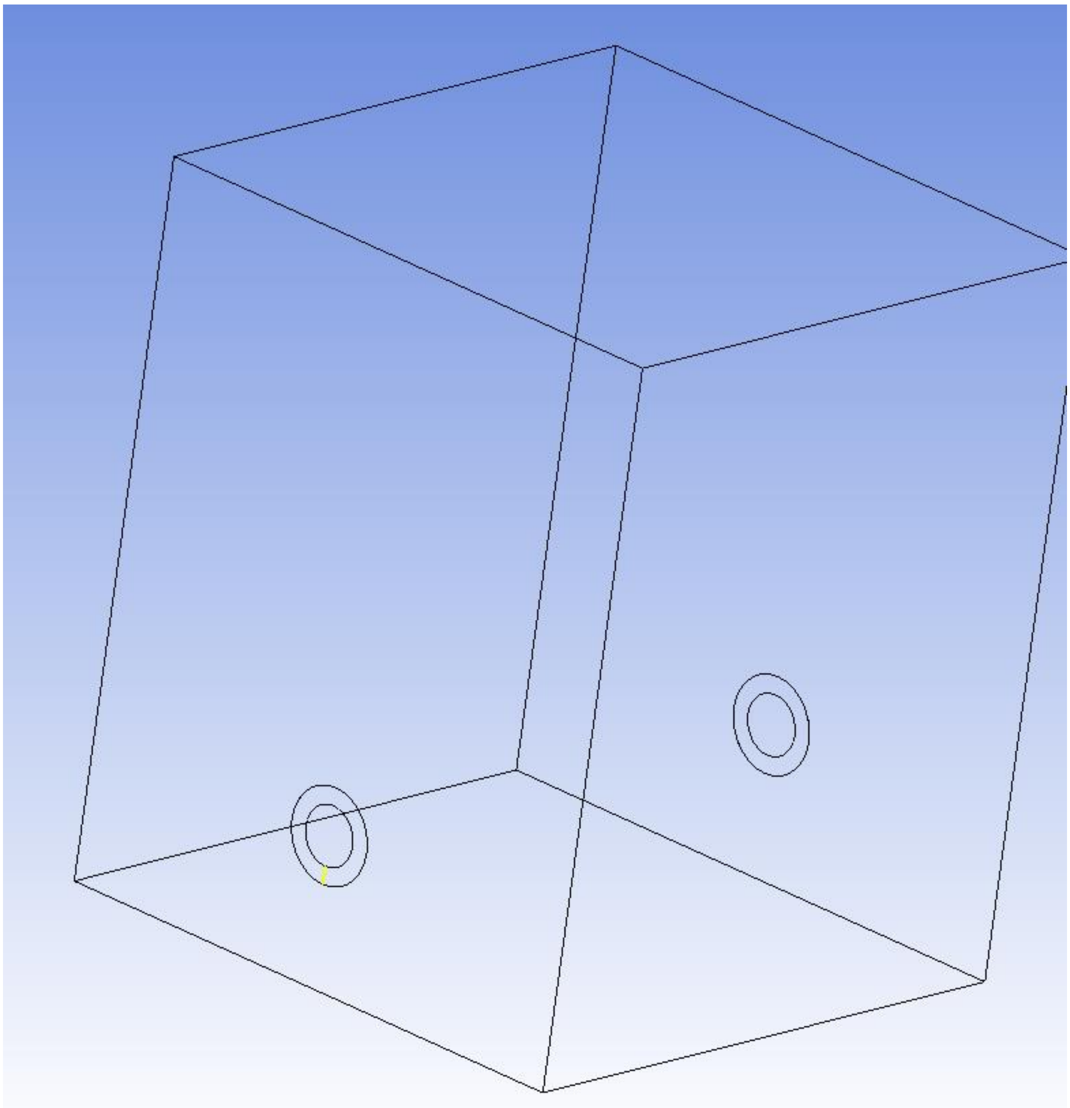




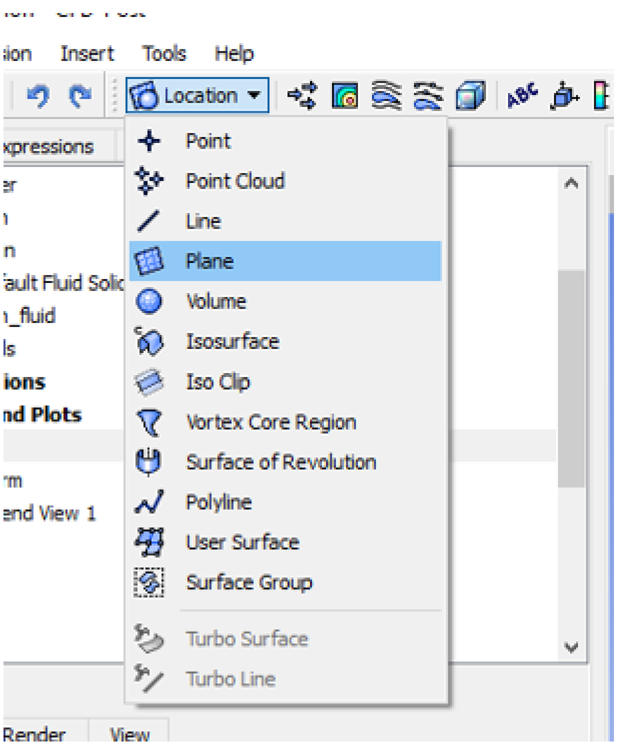
4) Turn off contours and close *Timestep Selector*



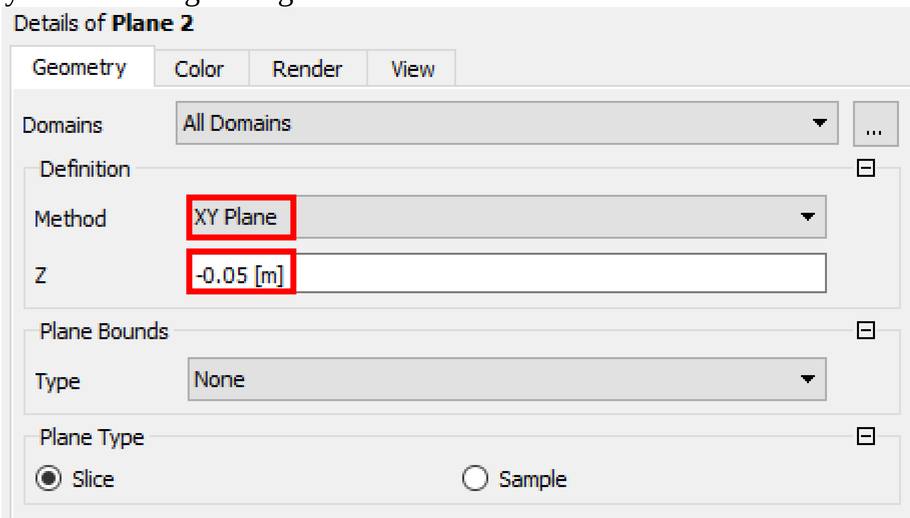
5) Rotate the model to view as below



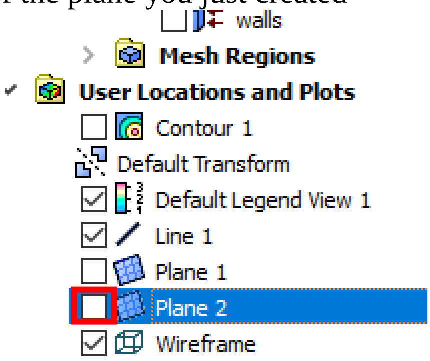
6) Create a transverse plane with *Location->Plane*



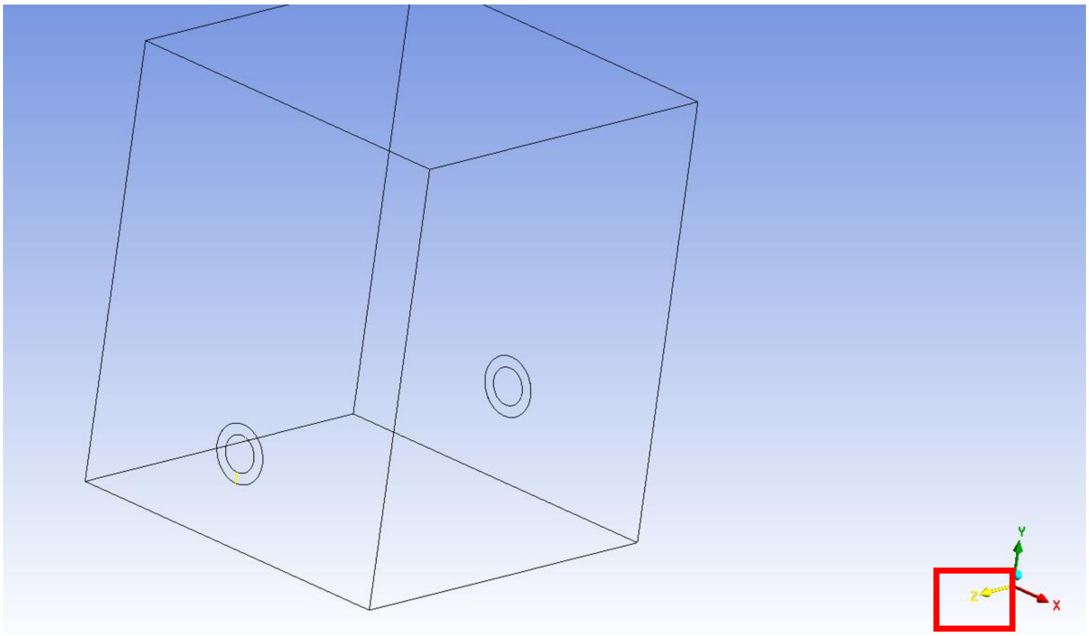
7) Apply the following settings



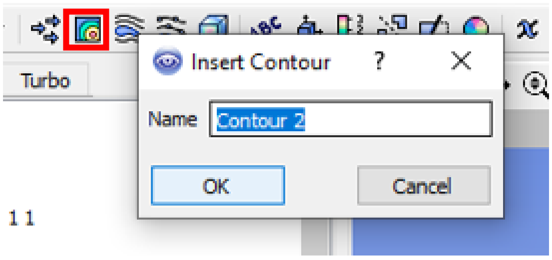
8) Turn off the visibility of the plane you just created



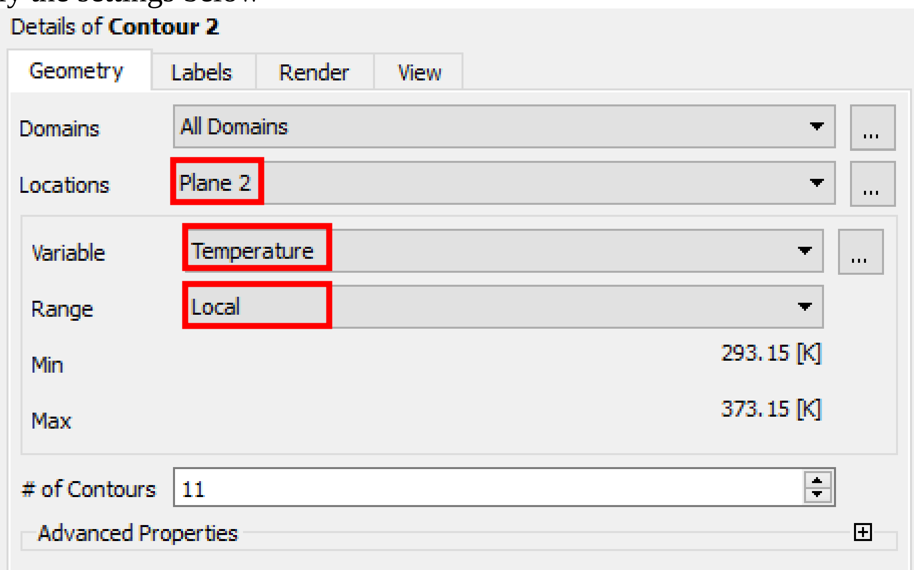
9) LMB press the Z axis



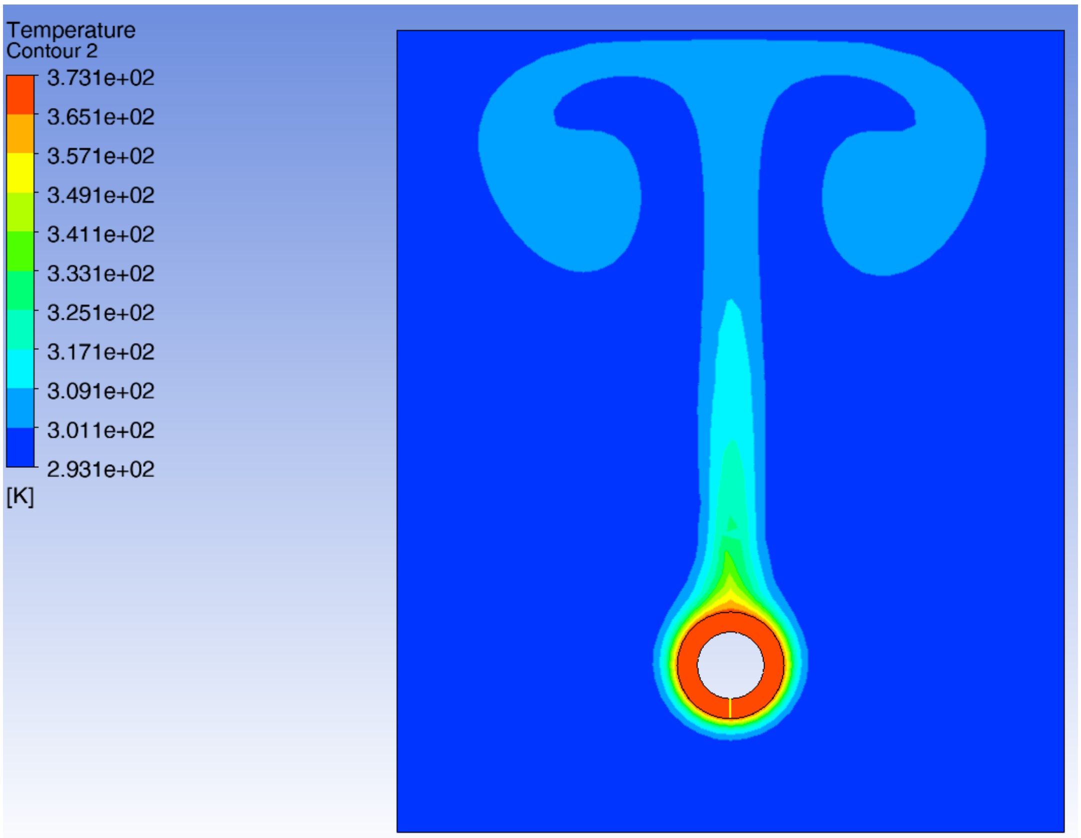
10) Create new contours



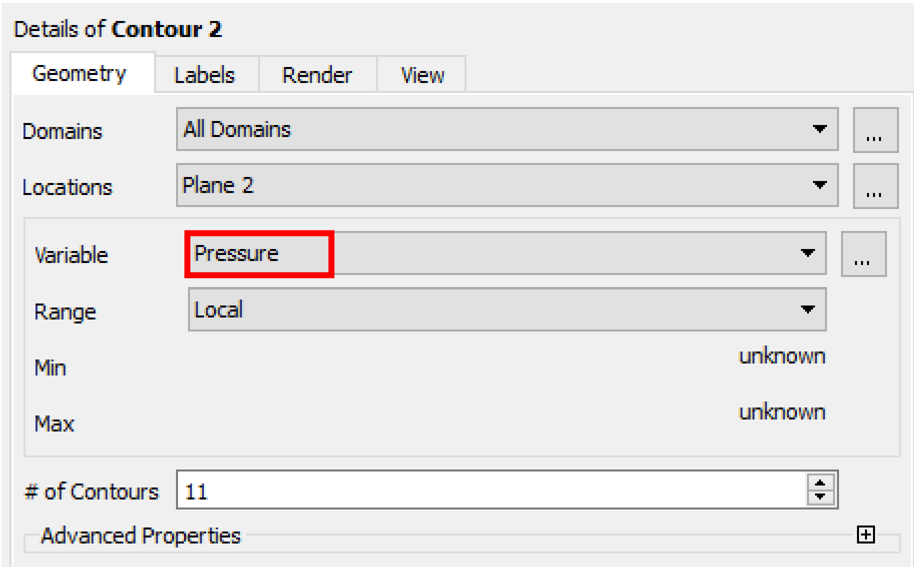
11) Apply the settings below



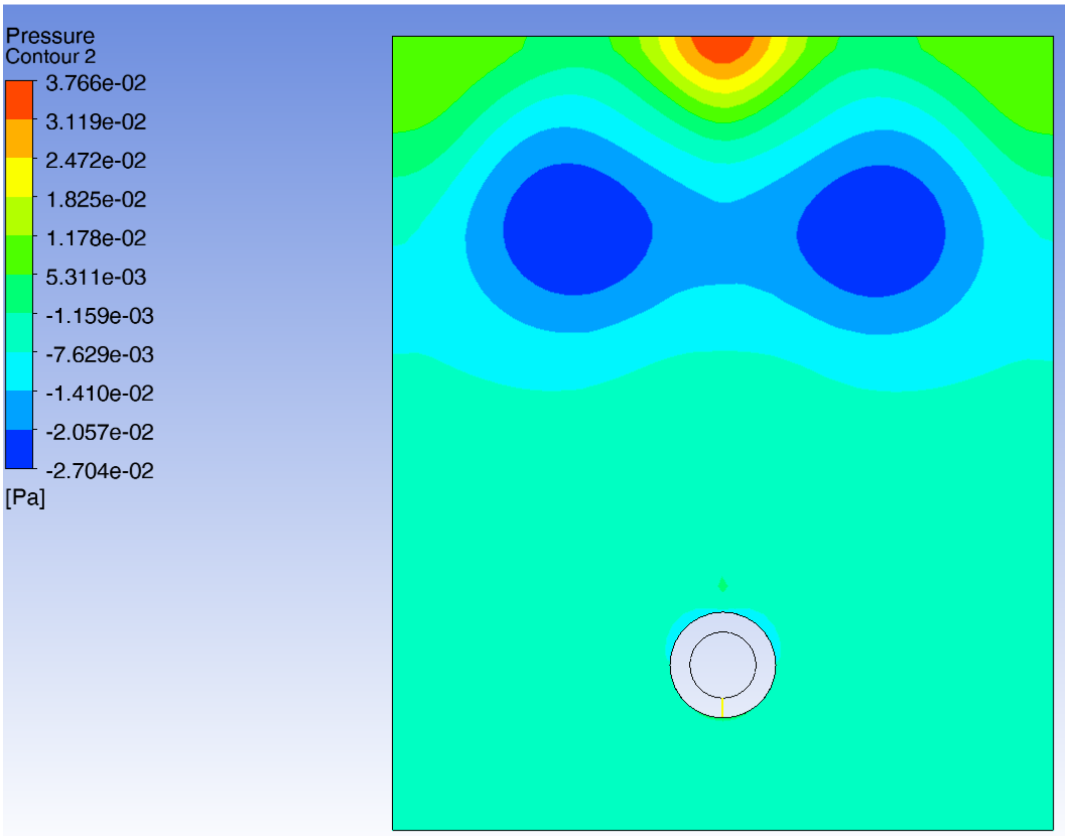
12) Using *Timestep Selector* observe temperature distributions at different times



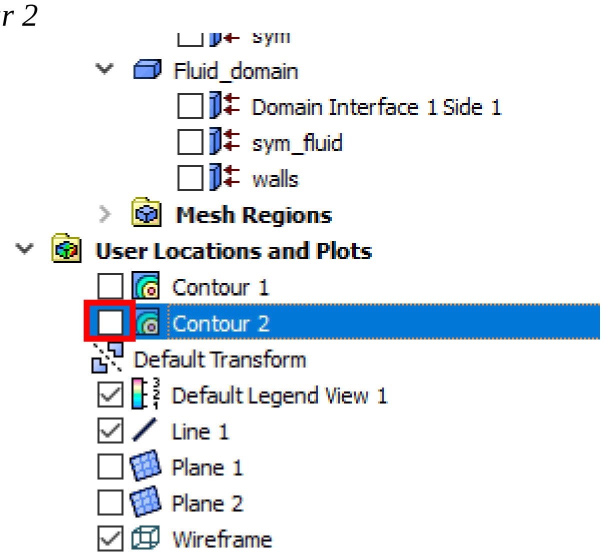
13) Edit *Contour 2* as follows



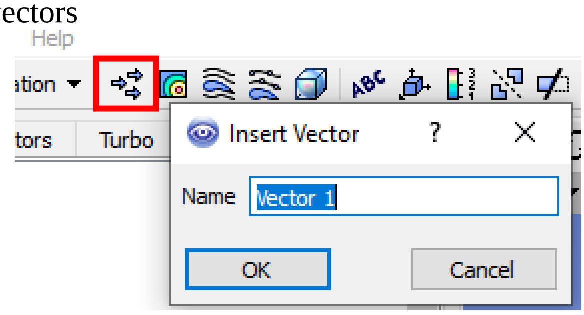
14) Using *Timestep Selector* observe pressure distributions at different times



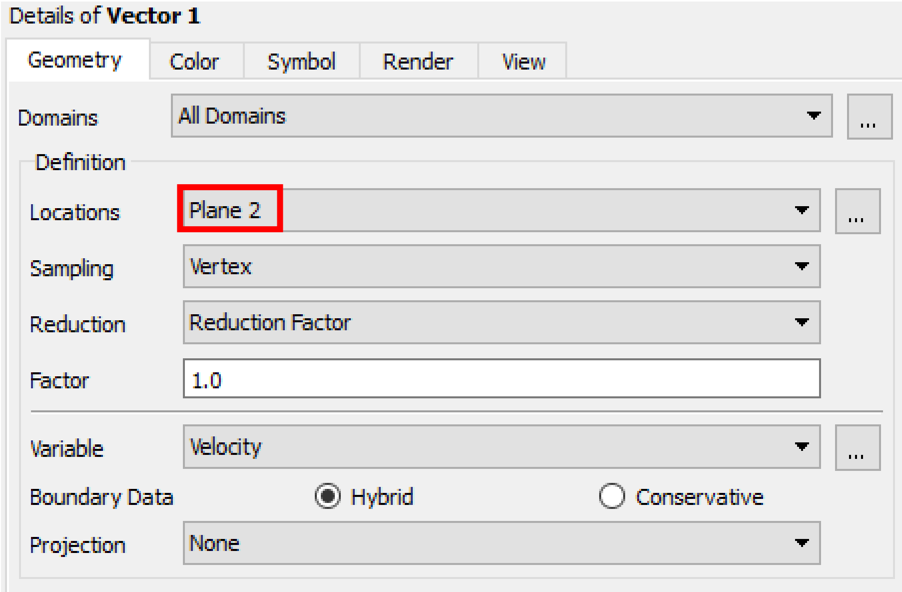
15) Turn off *Contour 2*



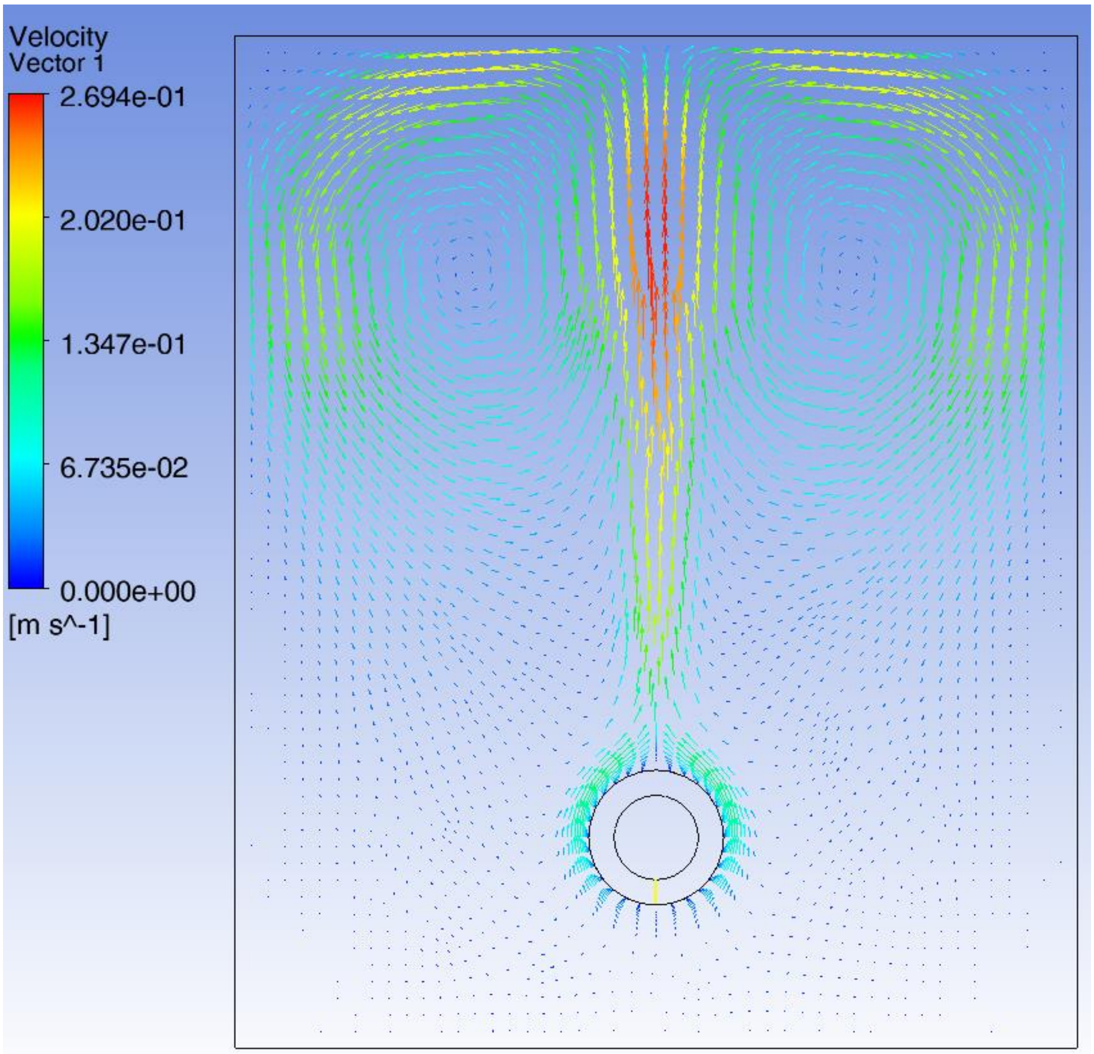
16) Create velocity vectors



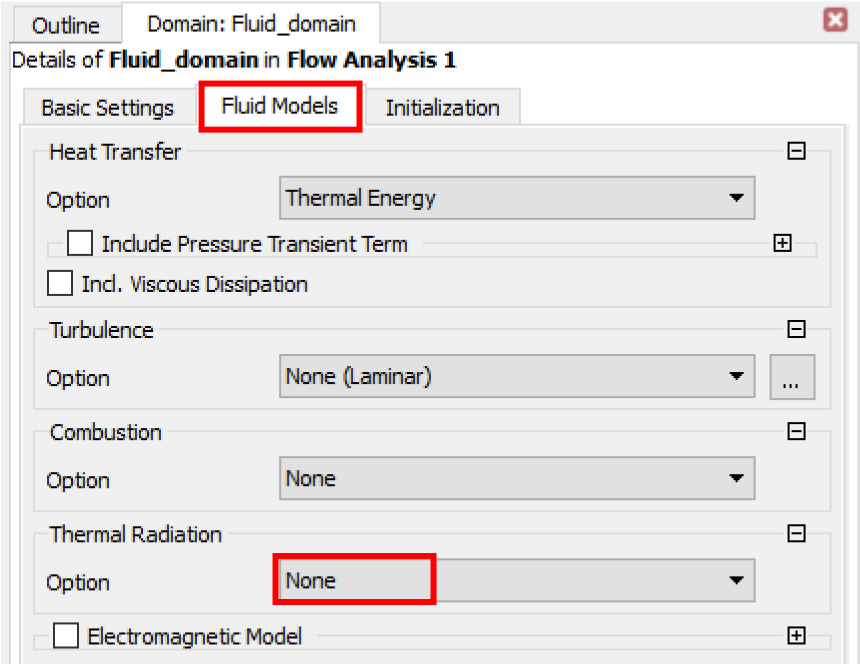
17) Apply the settings below and confirm *Apply*.



18) Using *Timestep Selector* observe the distribution of velocity vectors at different times



19) Self-help assignment: copy *RadiationConvection* and name it *onlyConvection.* Edit *Setup* in *onlyConvection* excluding heat radiation in the fluid domain. Repeat calculations and compare results with *RadiationConvection.*



20) Assignment to be performed alone: as in *Exercise no. 1* calculate the heat transfer rate on the pipe surface for time t = 2 s. Use the following *expression*:

=areaInt(Wall Heat Flux)@Domain Interface 1 Side 1

Results to be included in the report:

1) Contours of temperature distribution in a plane perpendicular to the axis of the pipe taking into account only convection and taking into account convection and radiation for times: 0; 0,1; 0,2; 0,3; 0,4; 0,5; 1,0; 2,0 s
2) Distribution of velocity vectors in a plane perpendicular to the axis of the pipe taking into account only convection and taking into account convection and radiation for times: 0; 0,1; 0,2; 0,3; 0,4; 0,5; 1,0; 2,0 s
3) Distribution of stremlines in a plane perpendicular to the axis of the pipe taking into account only convection and taking into account convection and radiation for times: 0; 0,1; 0,2; 0,3; 0,4; 0,5; 1,0; 2,0 s
4) Present in the table the heat transfer rates for all calculated cases.