



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

<b>1. Introduction .....</b>	<b>2</b>
<b>2. One-dimensional transient heat conduction through a cylindrical wall with taking into account convection and radiation.....</b>	<b>3</b>
2.1. Geometry .....	3
2.2. Numerical mesh .....	11
2.3. Symulacja numeryczna .....	25
<b>3. Results visualisation .....</b>	<b>41</b>

## 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.

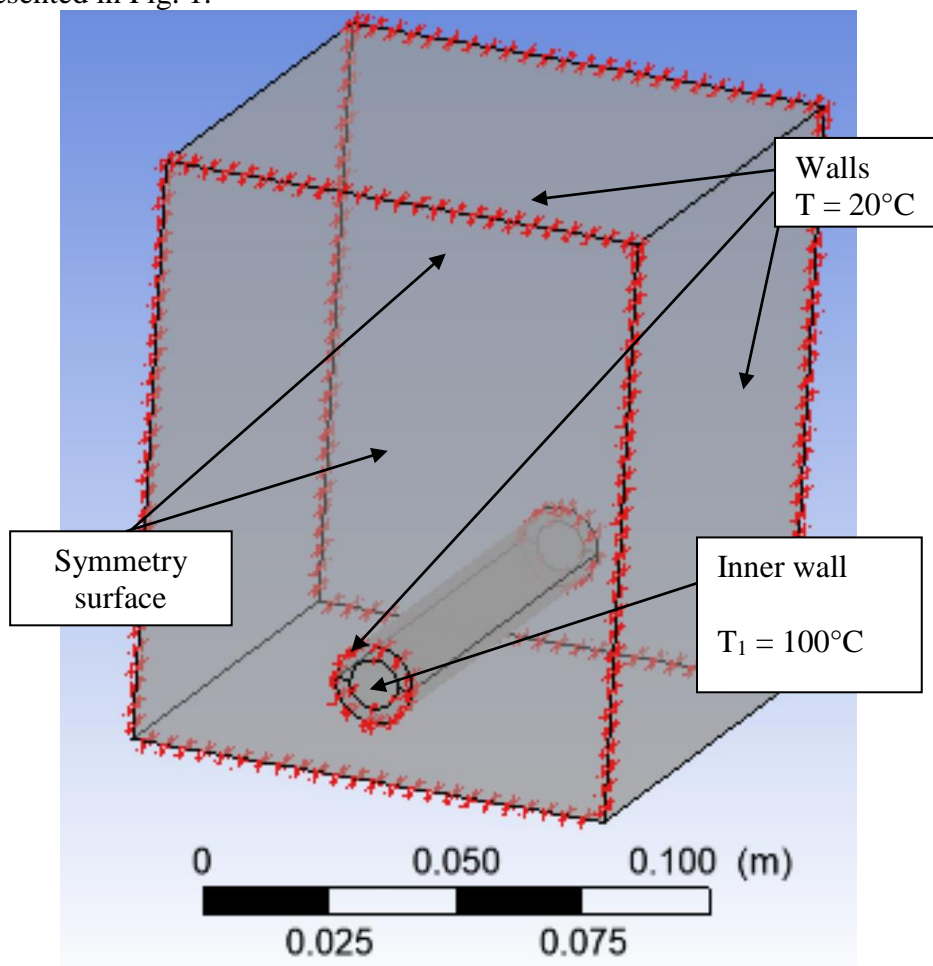


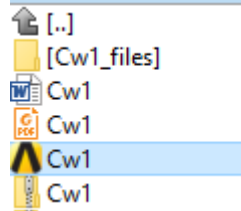
Fig. 1. Diagram of the issue of complex heat exchange

## 2. ONE-DIMENSIONAL TRANSIENT HEAT CONDUCTION THROUGH A CYLINDRICAL WALL WITH TAKING INTO ACCOUNT CONVECTION AND RADIATION

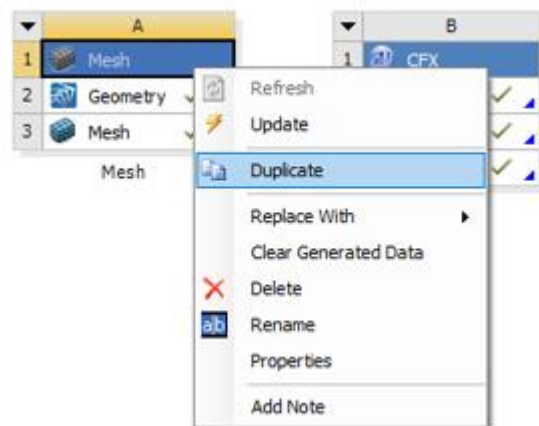
### 2.1. GEOMETRY

Do the following:

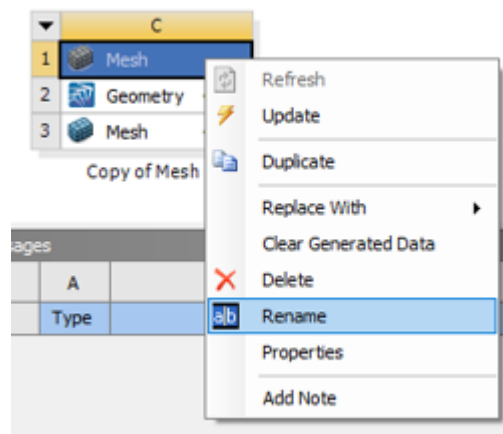
- 1) Open *Workbench* file containing *Exercise no. 1*



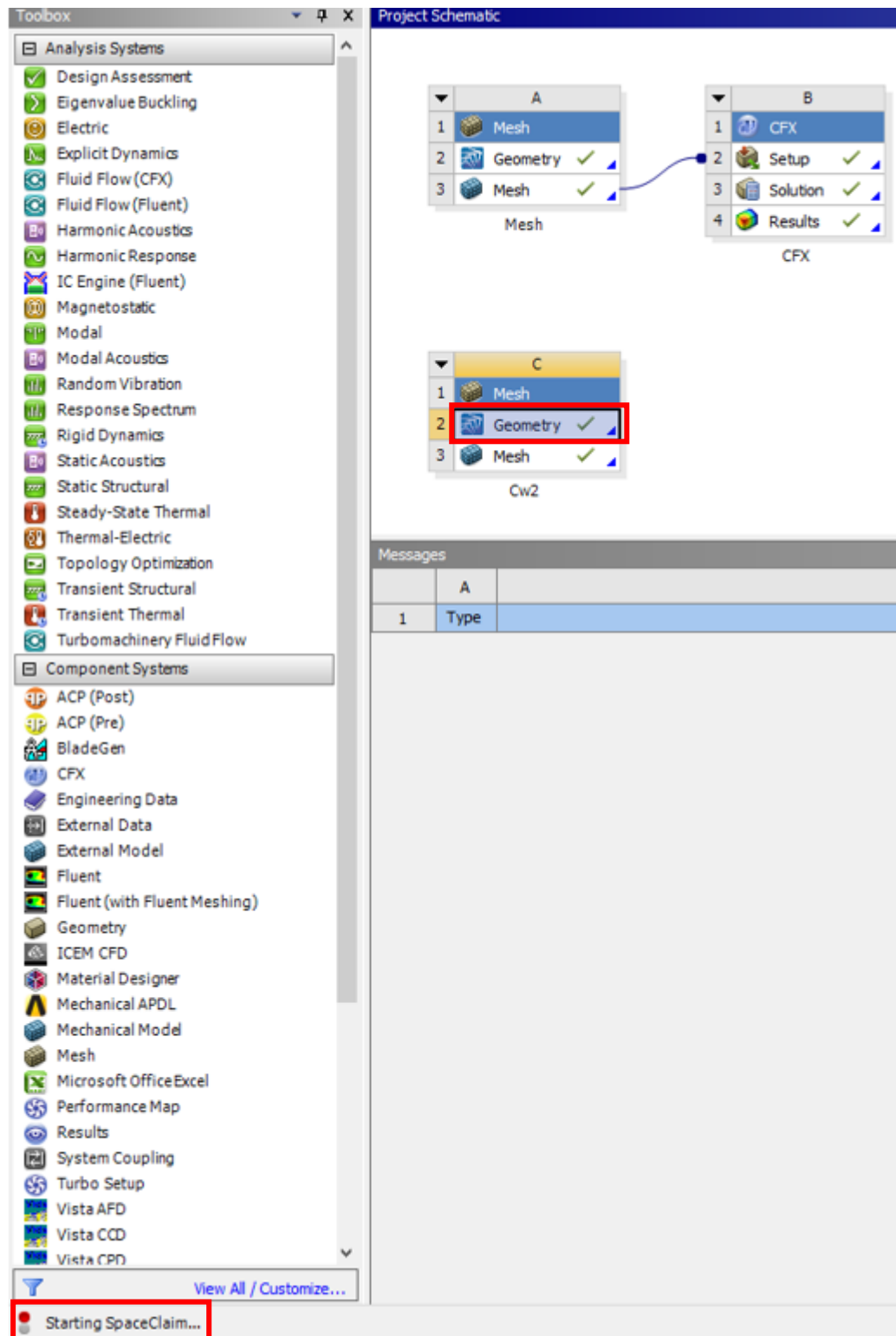
- 2) RMB click on *Mesh* and select *Duplicate* to copy geometry from *Exercise no. 1*



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



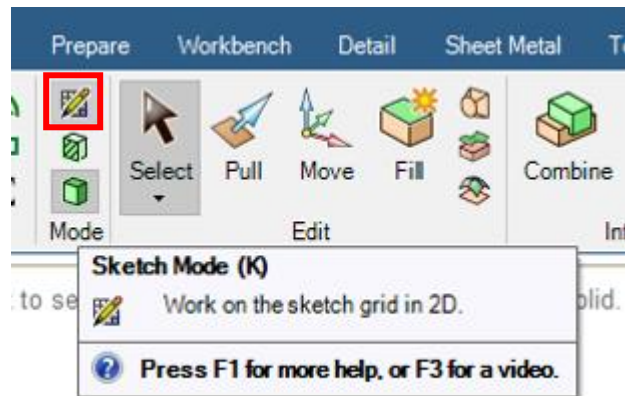
- 4) Double click LMB on *Geometry* in *Cw2*



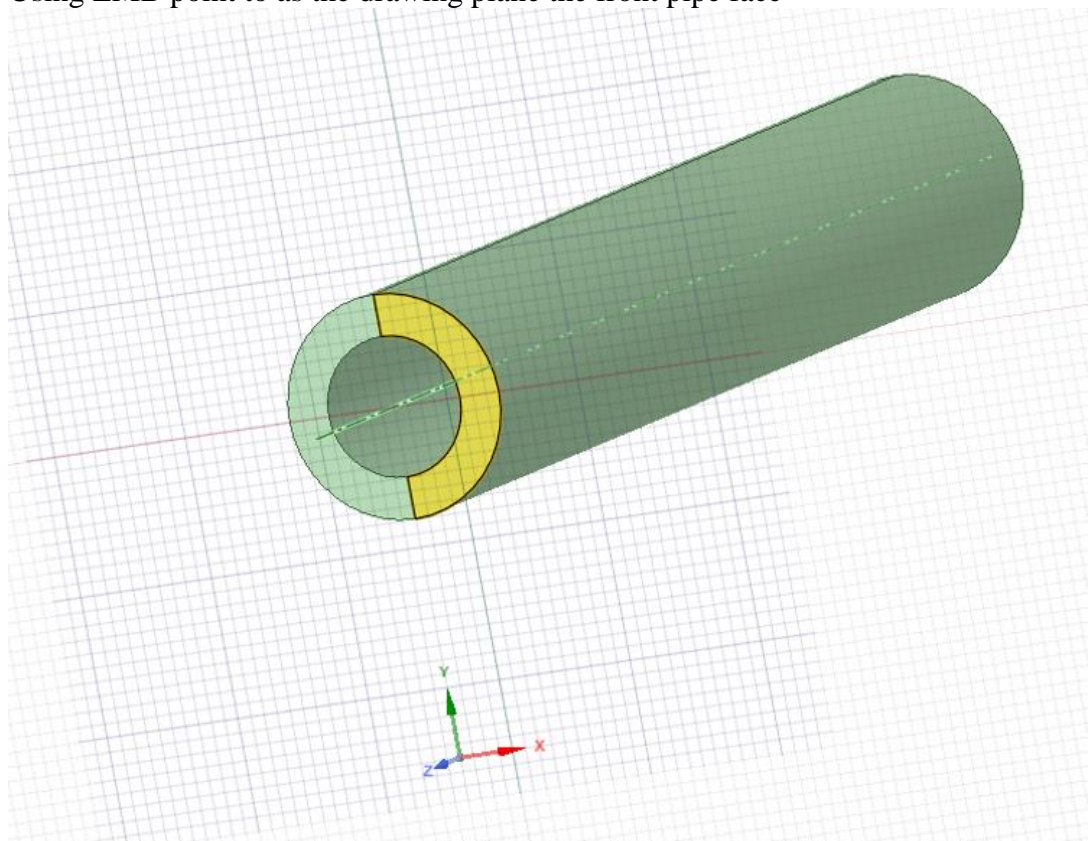
- 5) Turn off the visibility of the cutting plane from *Exercise no. 1*




6) Choose *Sketch Mode*

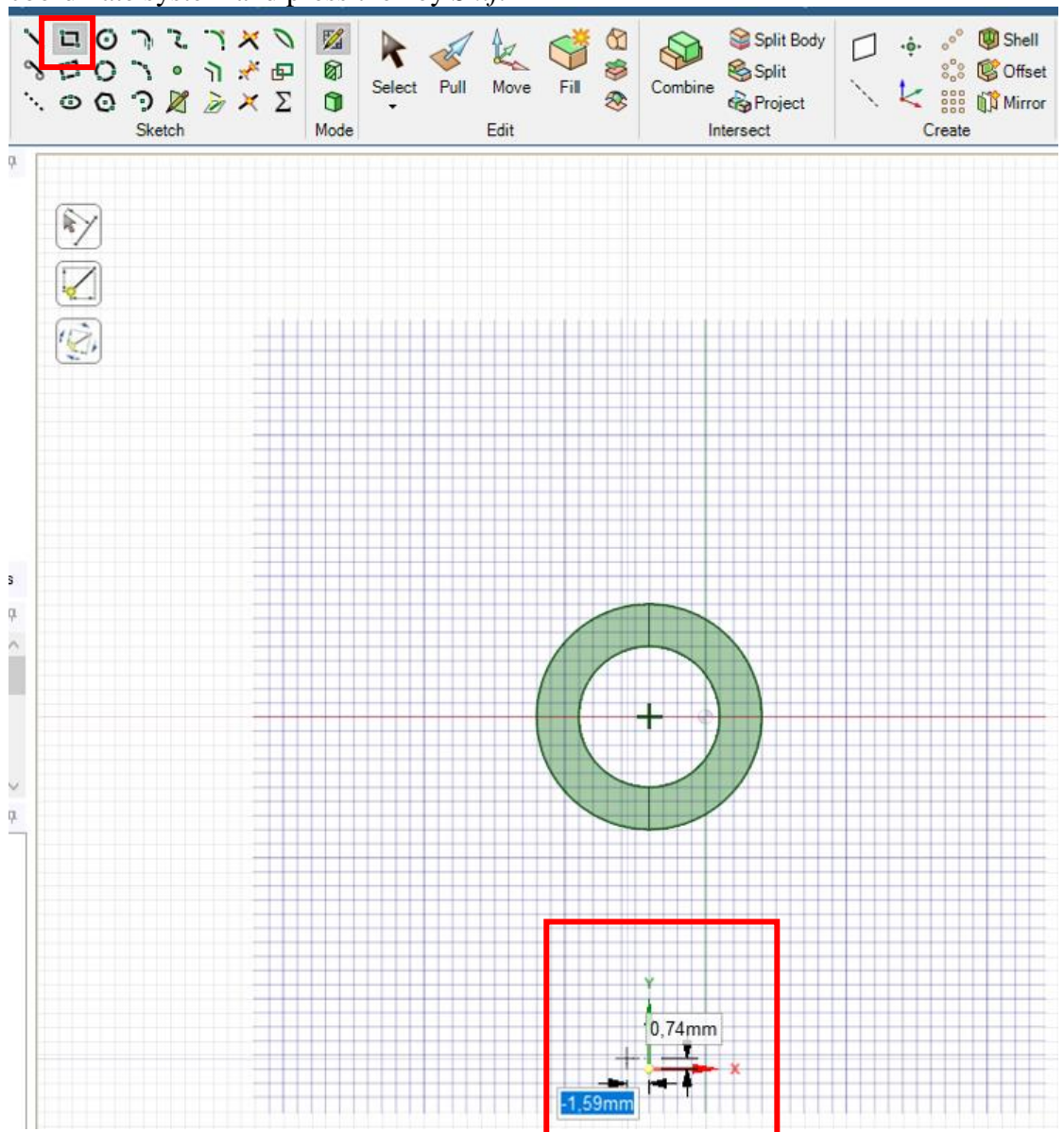


7) Using LMB point to as the drawing plane the front pipe face



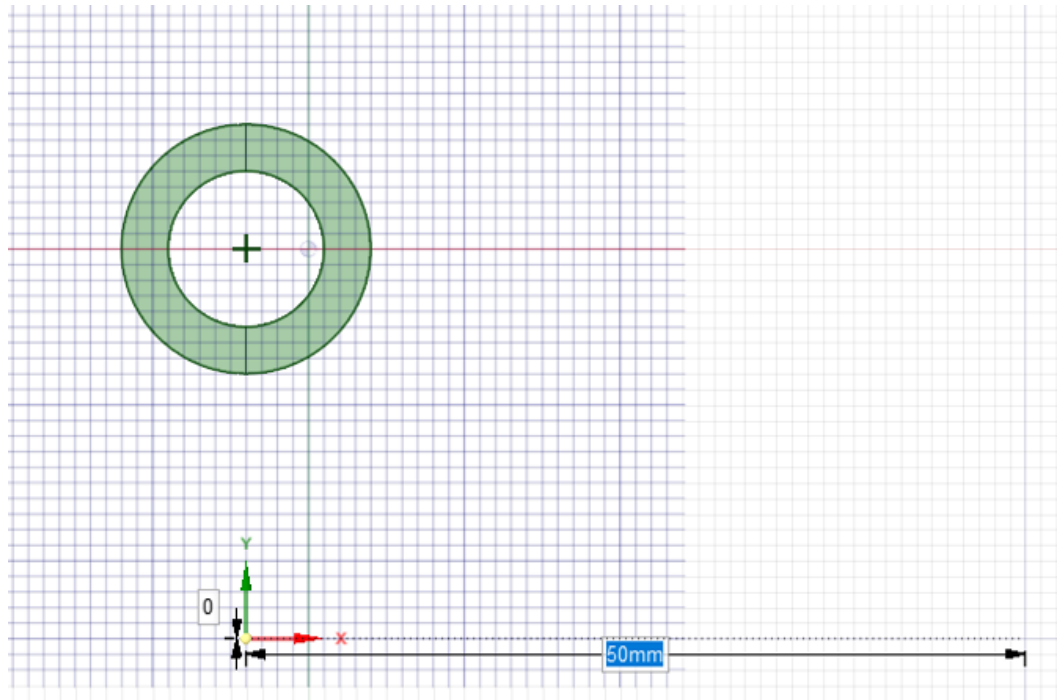
8) Set the view parallel to the screen (*Shift + v* lub )

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

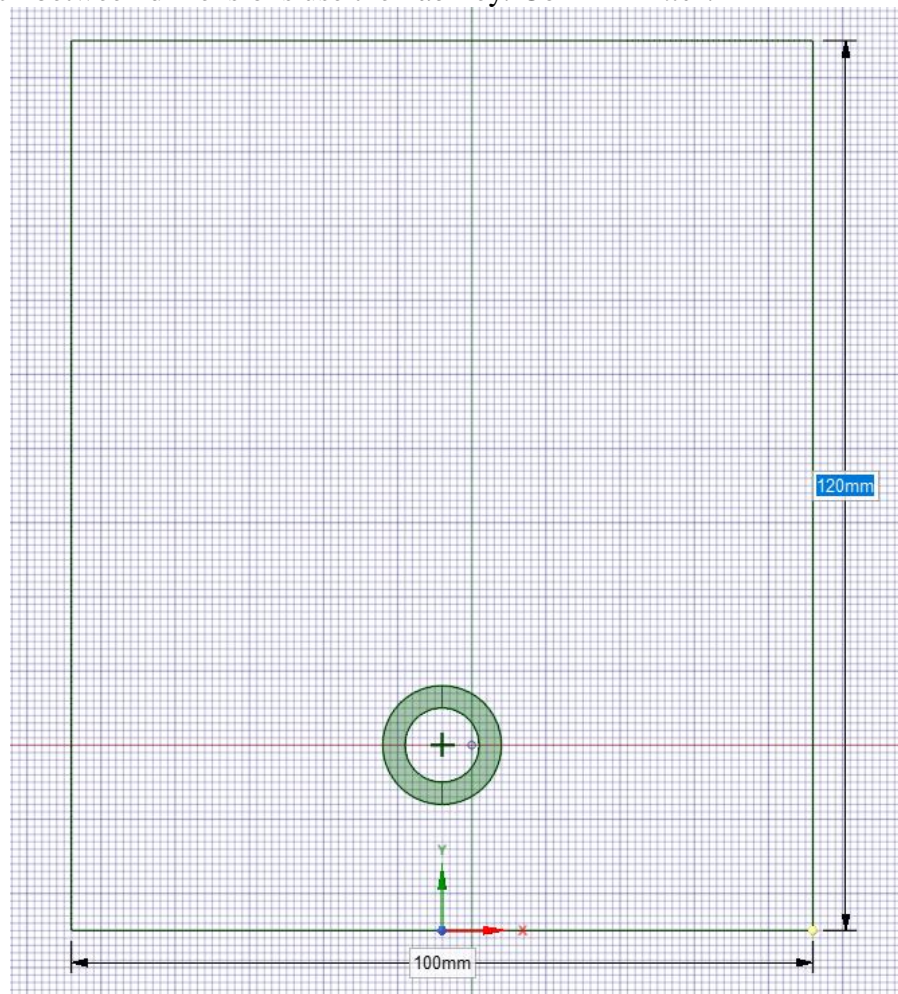


- 10) After *Shift* clicking two dimension edit fields will appear. Set horizontal dimension as 50 mm and vertical dimension as 0 mm. To switch between dimensions use the *Tab* key. Confirm *Enter*.

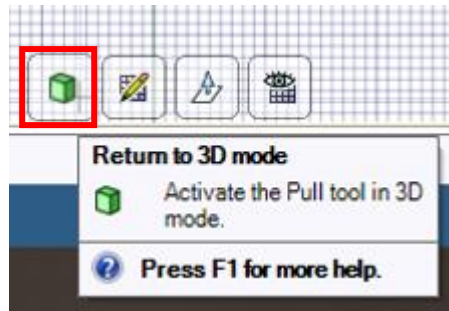




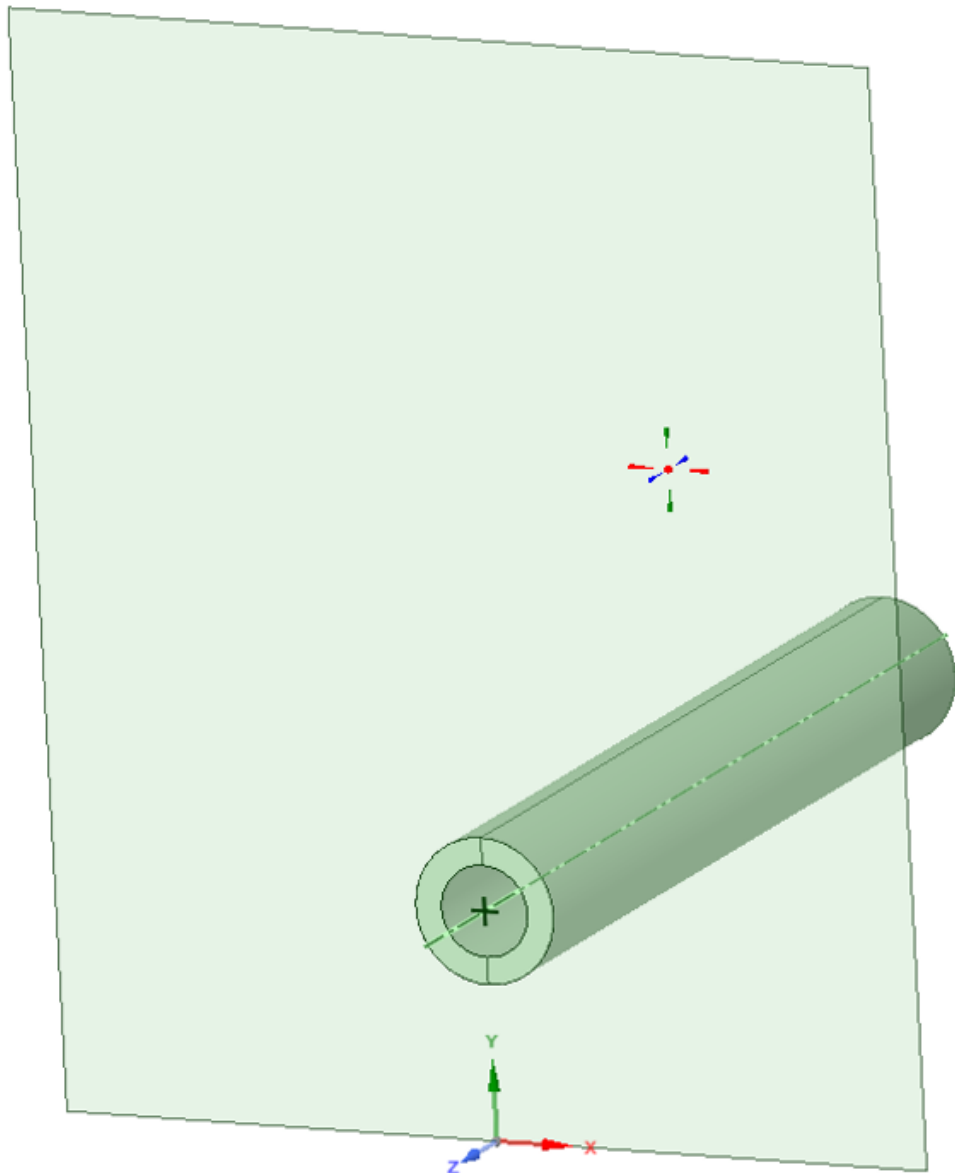
- 11) Set horizontal dimension to 100 mm and vertical dimension as 120 mm. To switch between dimensions use the *Tab* key. Confirm *Enter*.



- 12) Return to 3D view

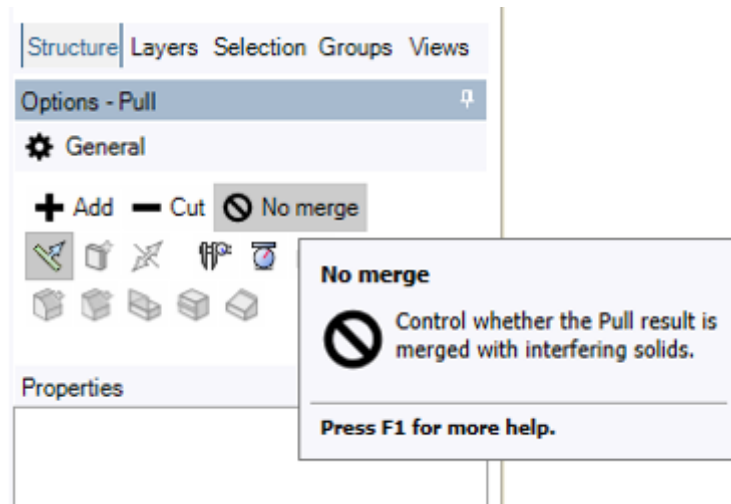


13) By sliding the *Scroll* button pressed rotate the view as below

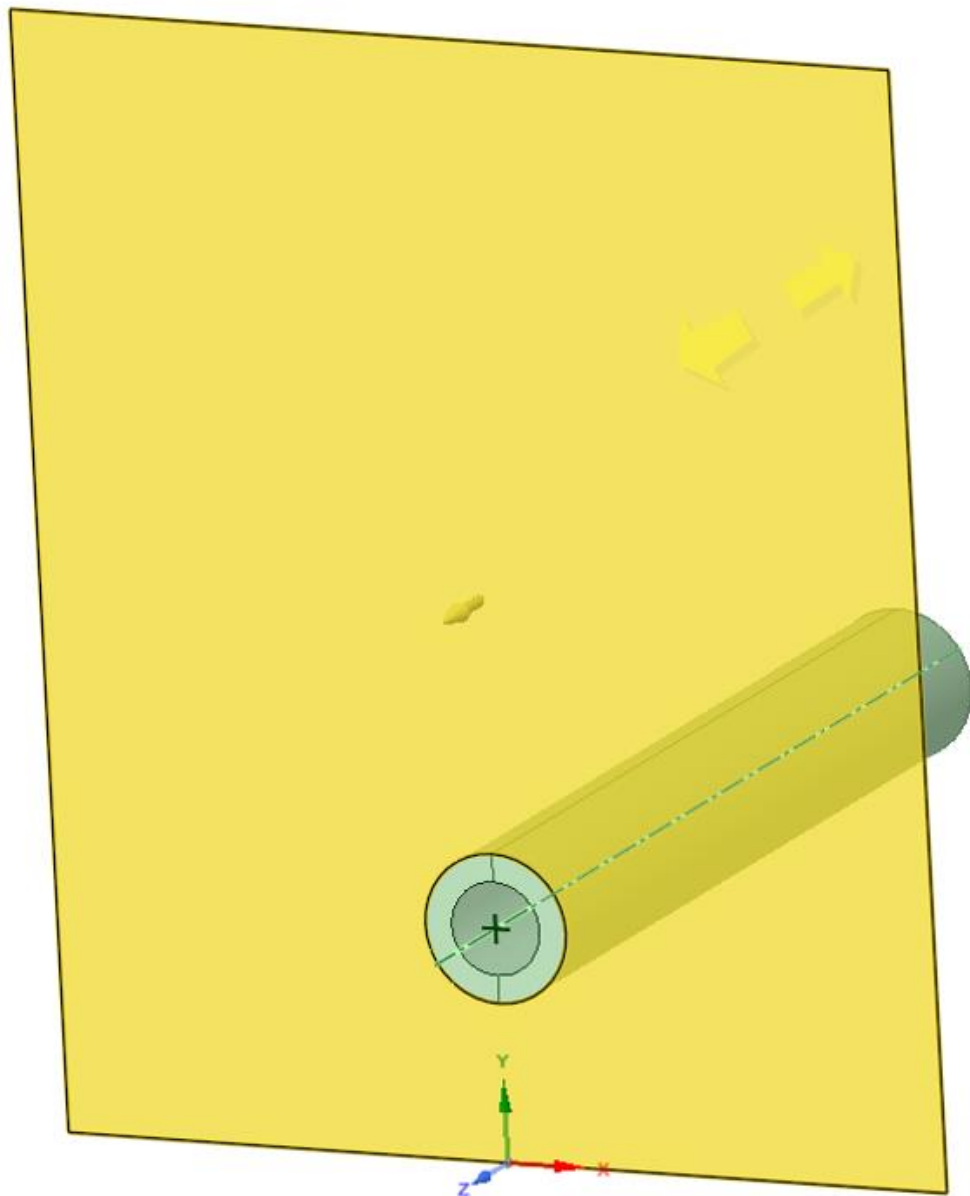


14) On the left side of the screen, with LMB select *No merge*

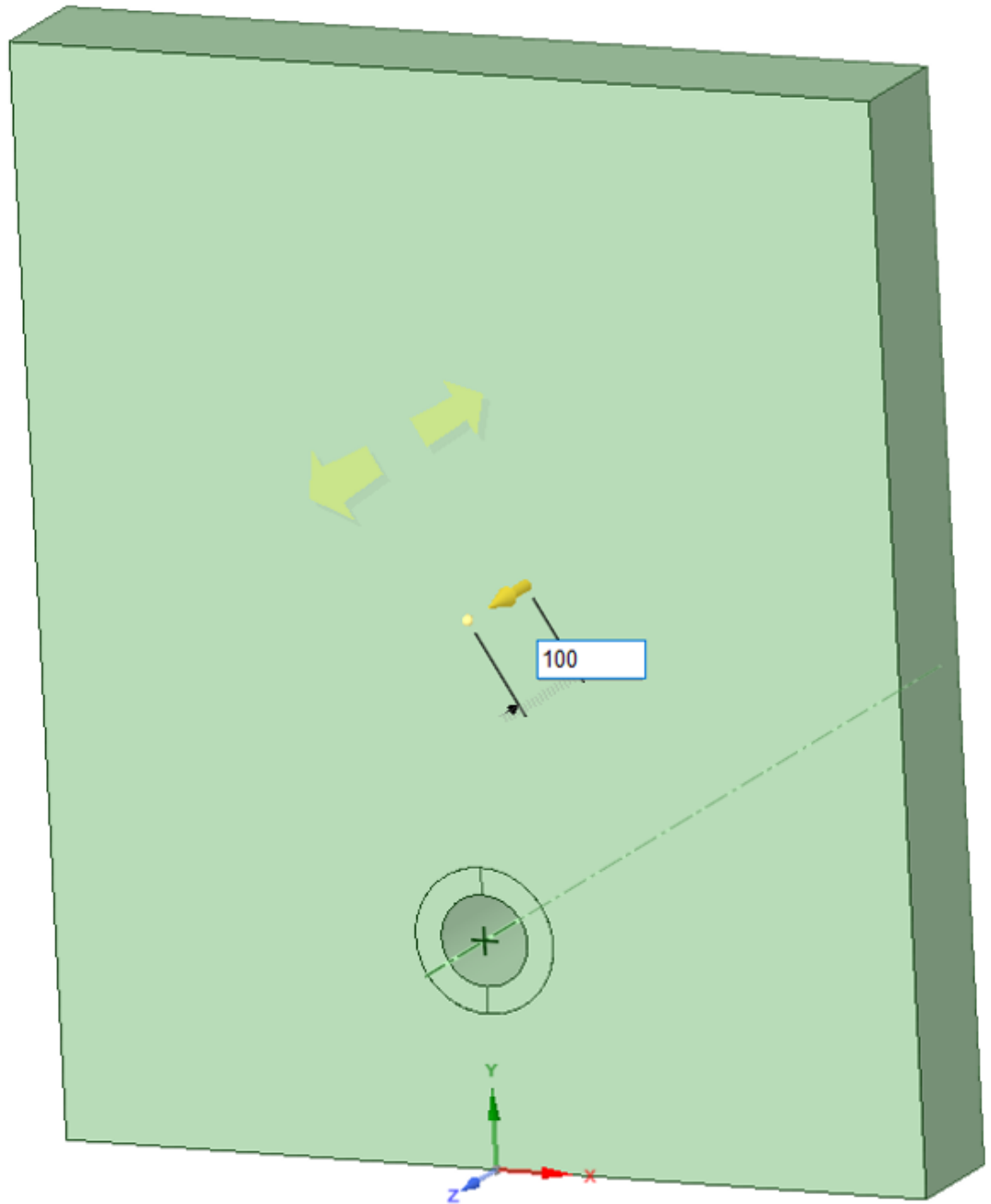




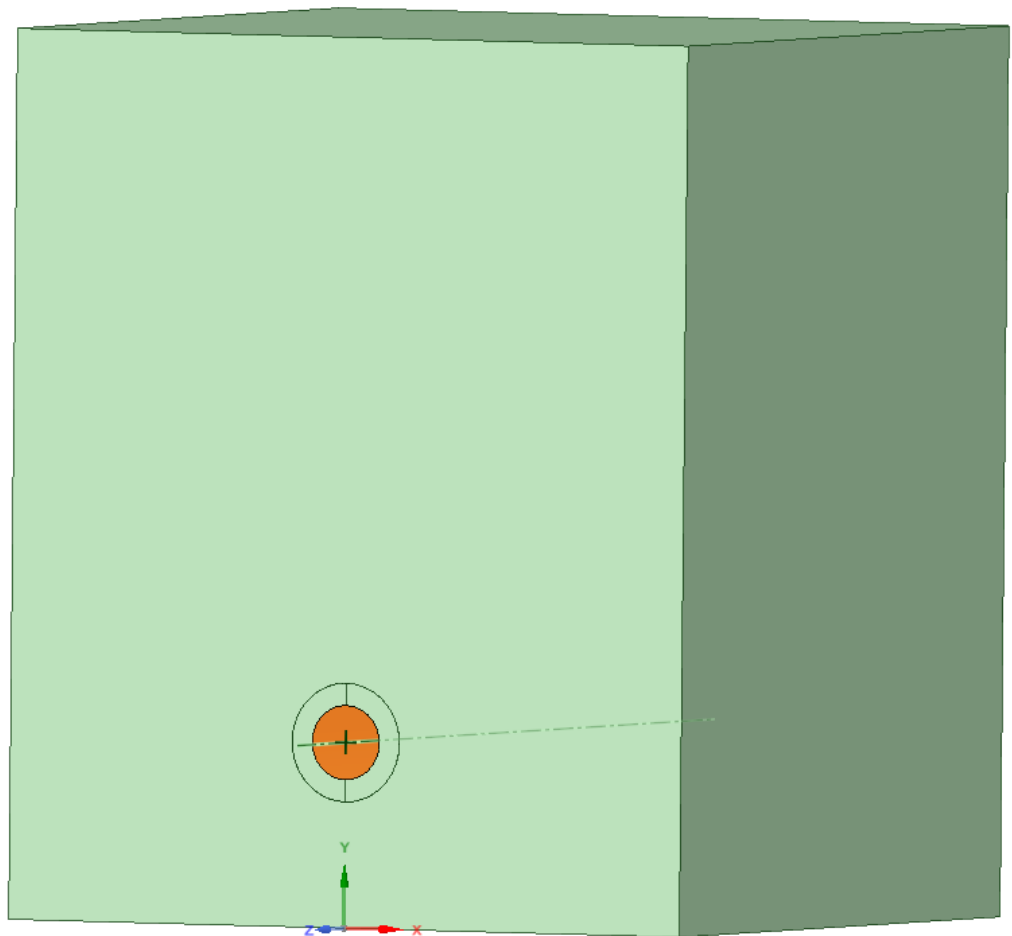
15) Then position the cursor so that the yellow arrows show up



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



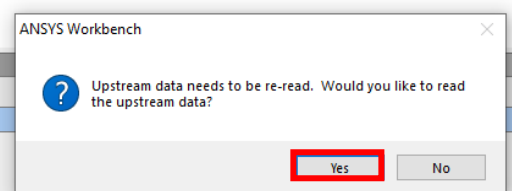
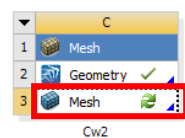
- 17) Press *Esc* to go out from *Pull* and select with LMB pipe inlet surface



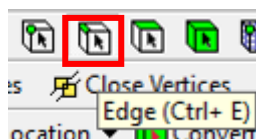
- 18) Press *Delete* to remove the orange surface and close *Spaceclaim*.
- 19) Save project in *Workbench* (*Ctrl + s*).

## 2.2. NUMERICAL MESH

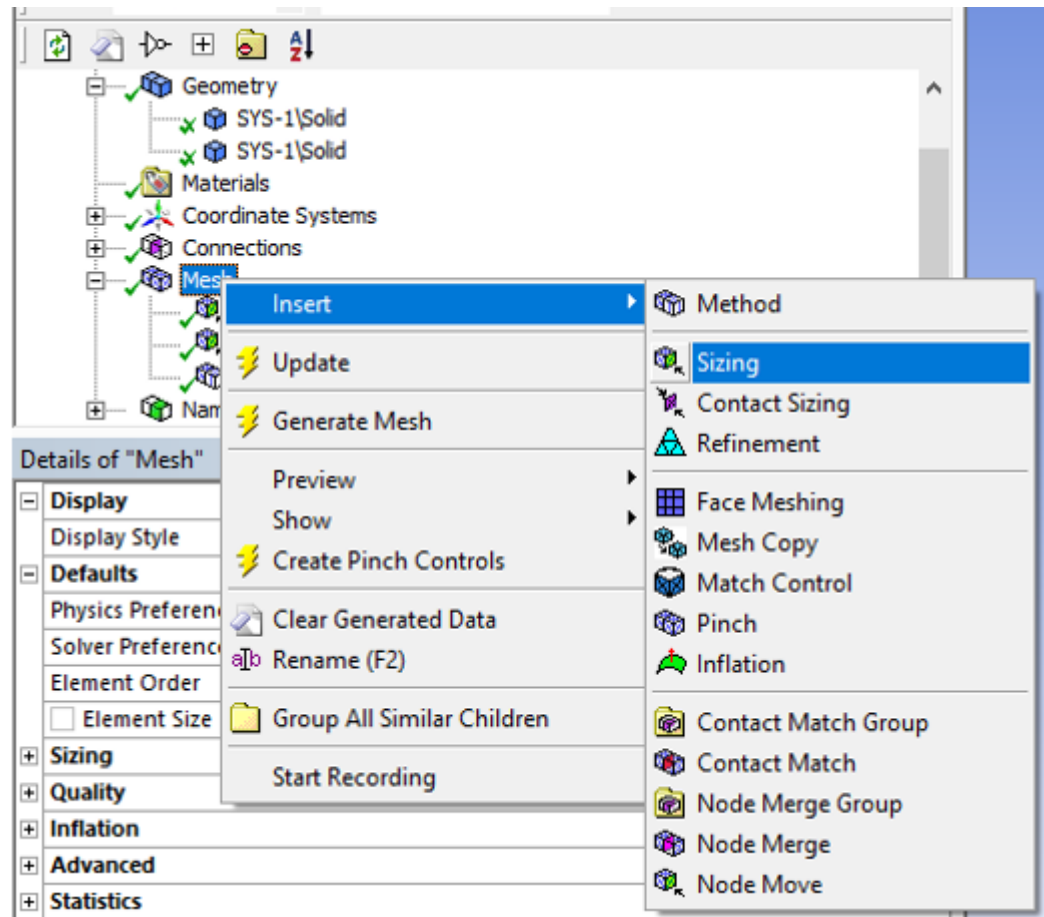
- 1) Double click on *Mesh* and select *Yes*



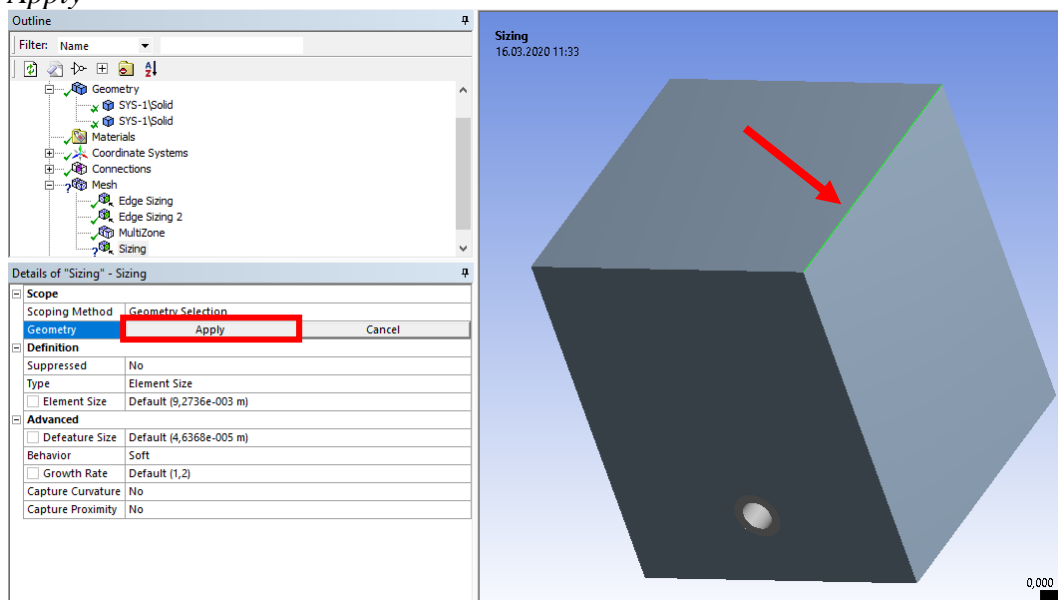
- 2) Select an edge selection filter



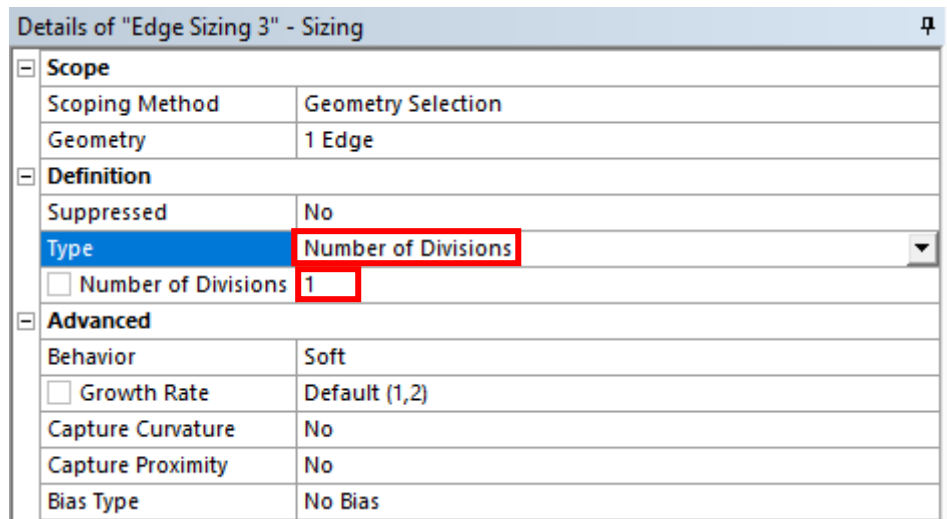
- 3) RMB on *Mesh* and choose *Insert->Sizing*



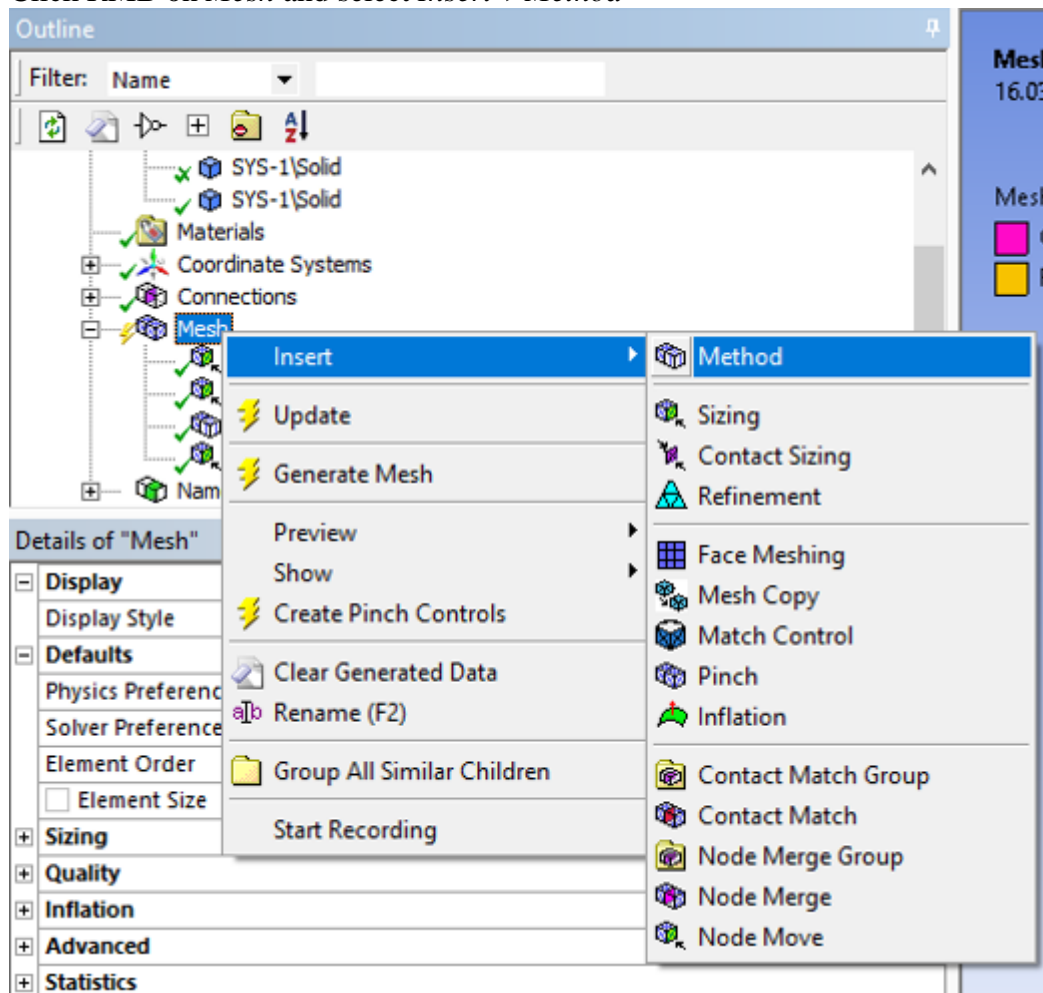
- 4) With LMB select the edge of the cuboid along the pipe direction, then confirm *Apply*



- 5) In *Details* apply the following settings

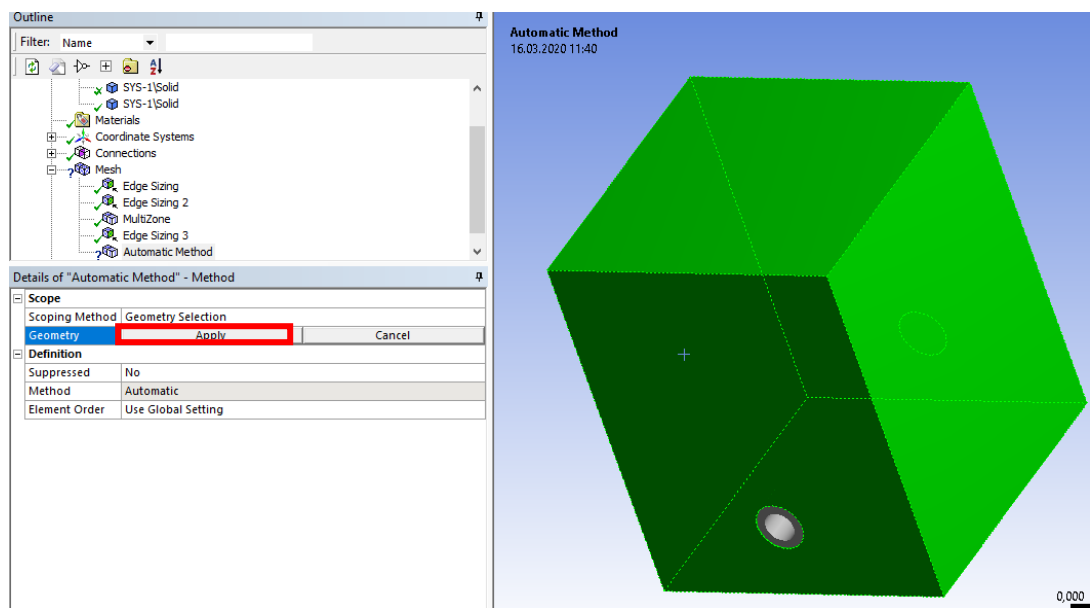


- 6) Click *Generate Mesh* and observe the numerical mesh.
- 7) Click RMB on *Mesh* and select *Insert->Method*



- 8) LMB select cuboid and confirm *Apply*.

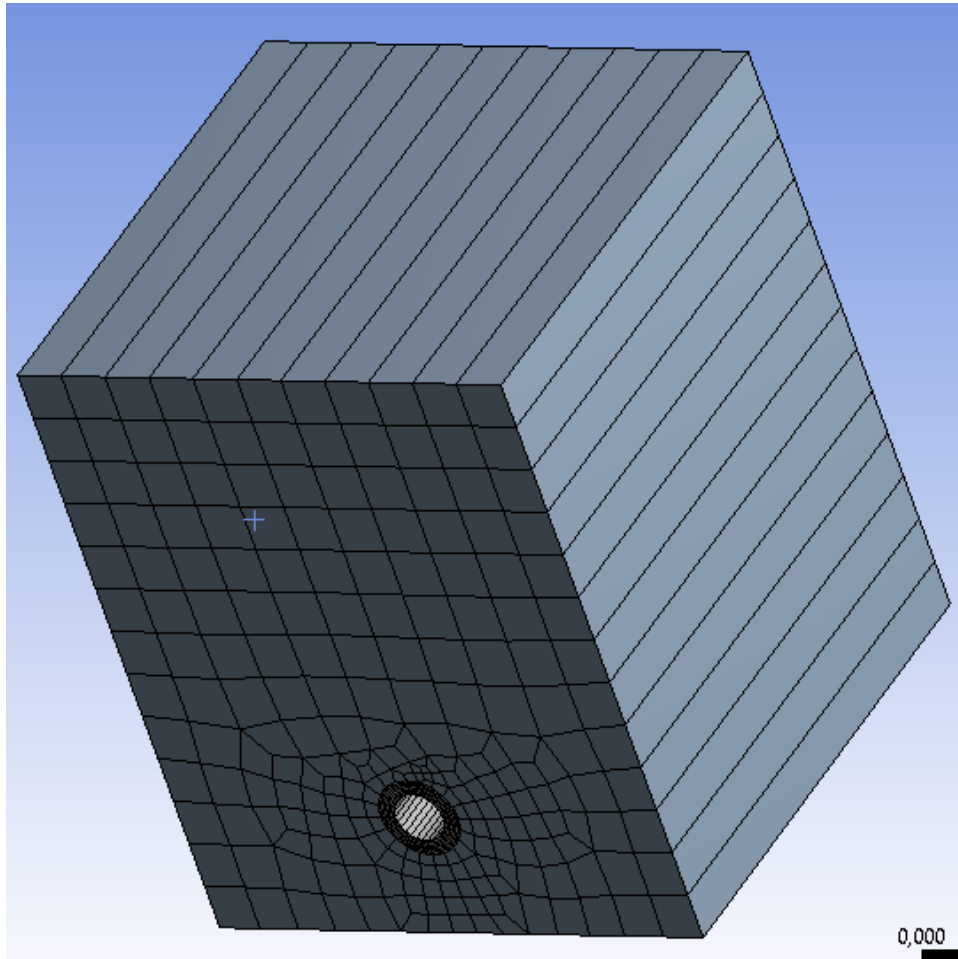




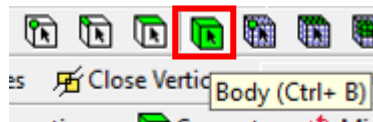
9) In *Details* apply the settings below

Details of "MultiZone 2" - Method	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Body
<b>Definition</b>	
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
<input type="checkbox"/> Sweep Element Size	Default
<b>Advanced</b>	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off
Minimum Edge Length	5,0265e-002 m
Write ICEM CFD Files	No

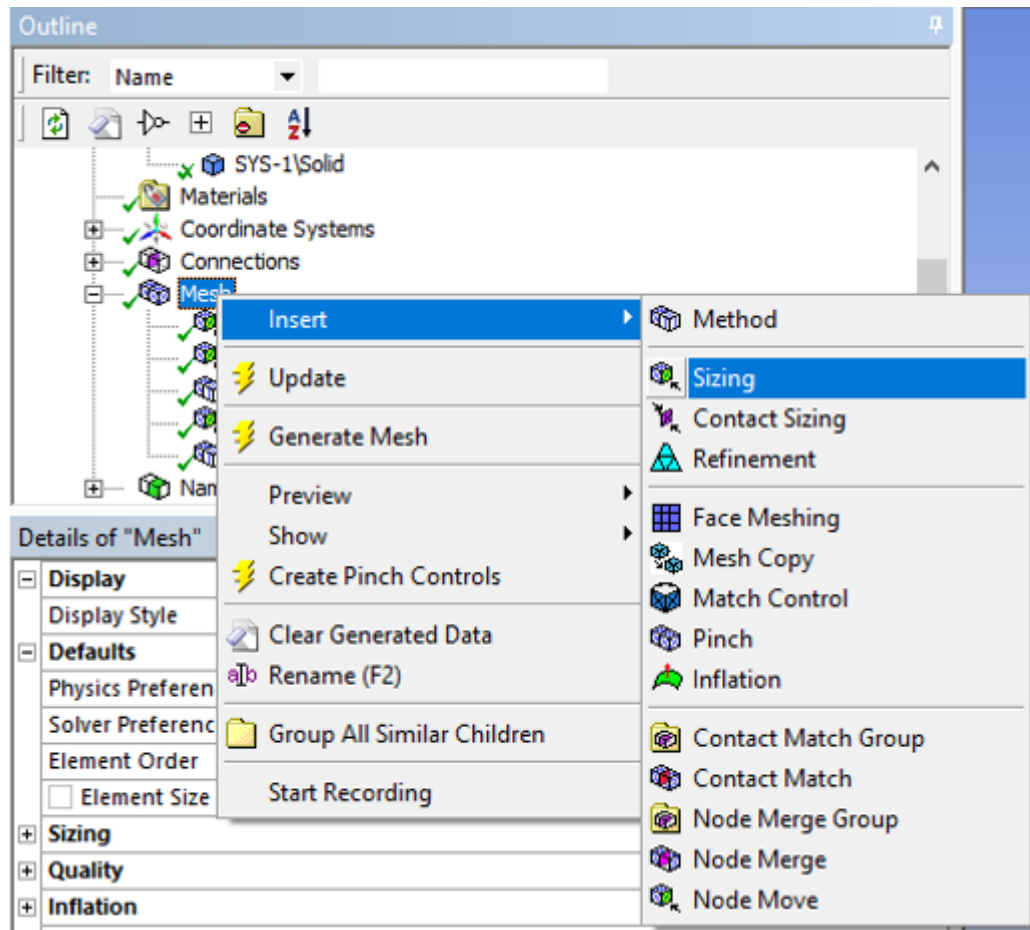
10) Click *Generate Mesh* and observe the numerical mesh.



11) Change filter into body selection

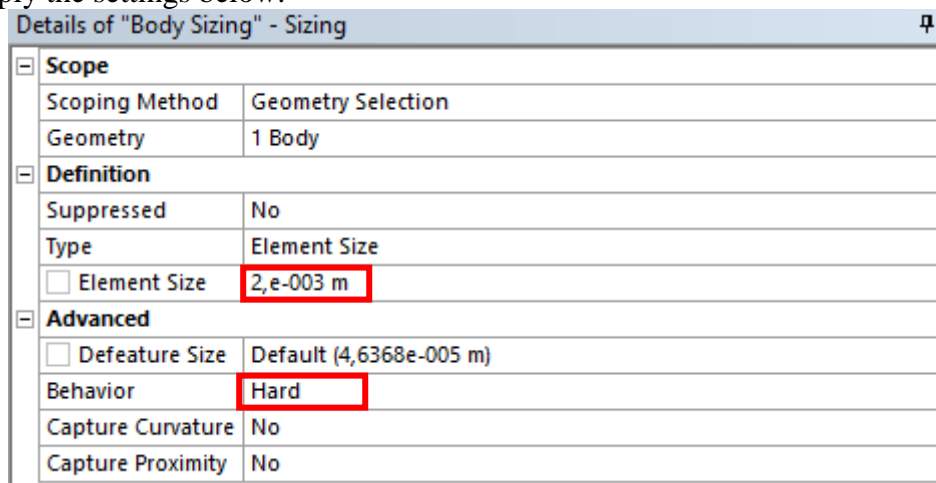


12) RMB on *Mesh* and select *Insert->Sizing*

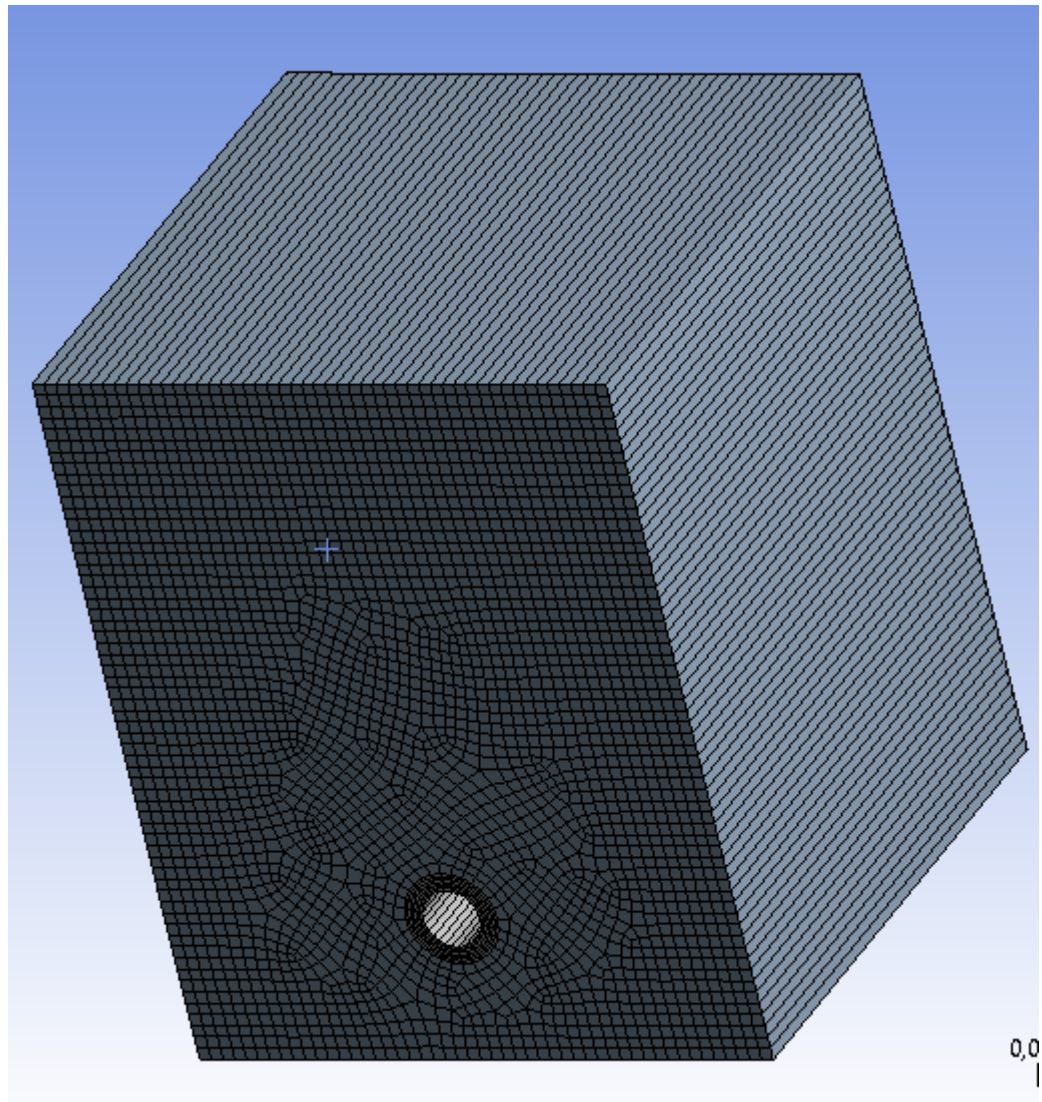


13) Select cuboid and confirm *Apply*.

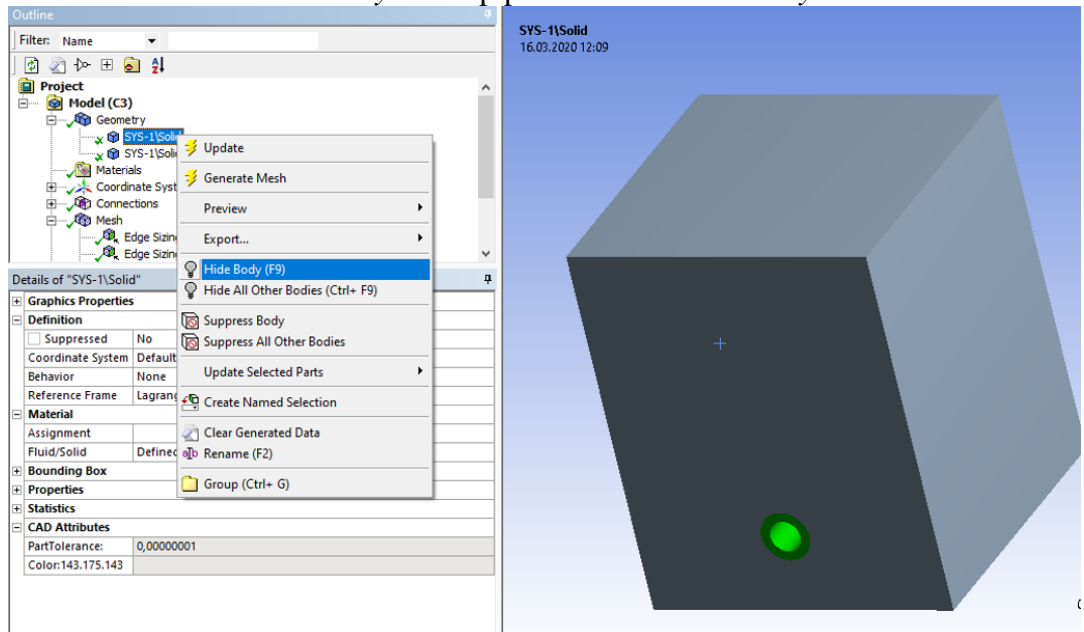
14) Apply the settings below.



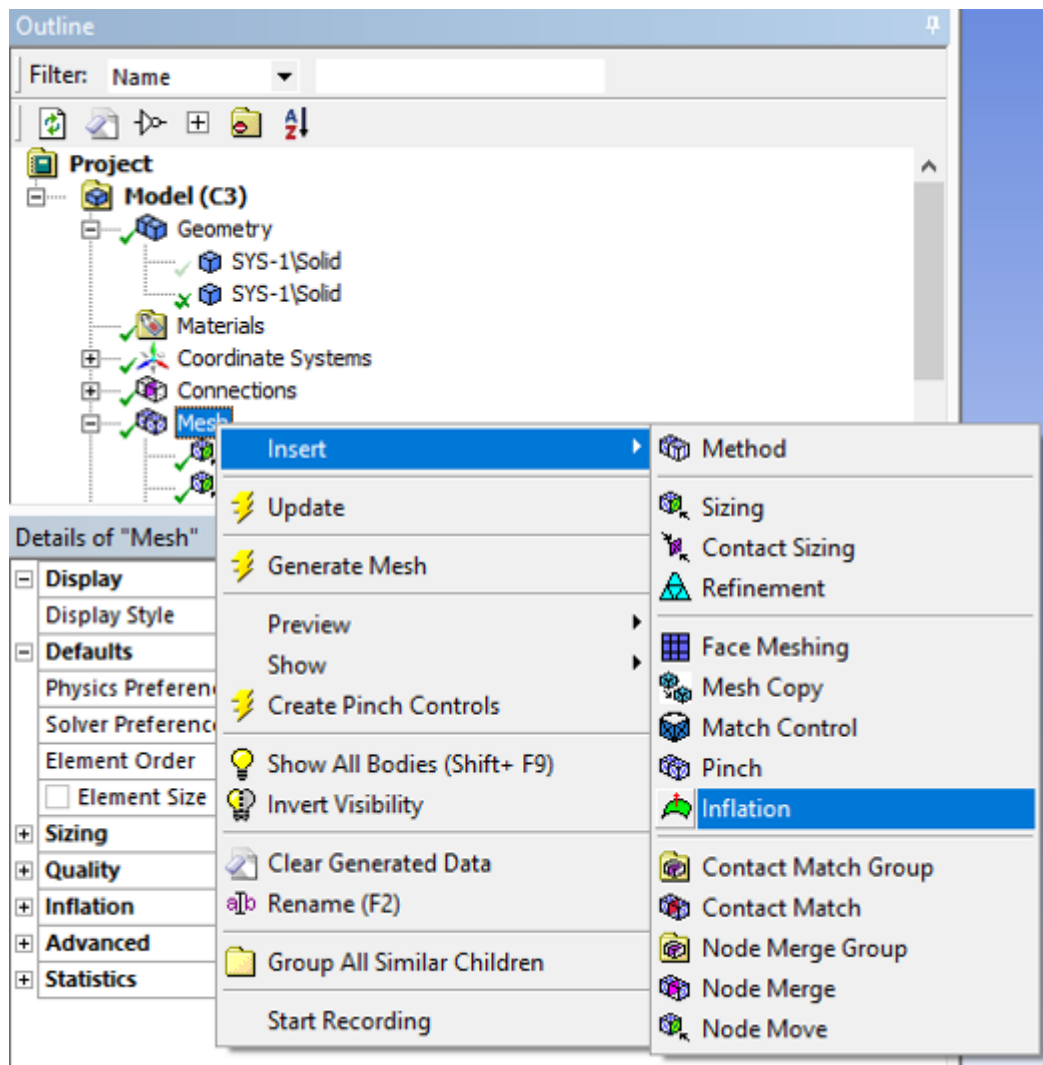
15) Click *Generat Mesh* and observe the numerical mesh.



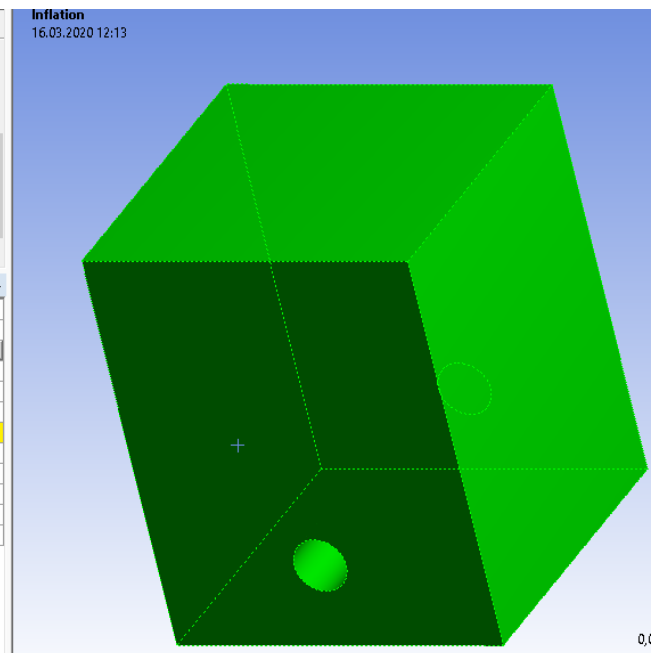
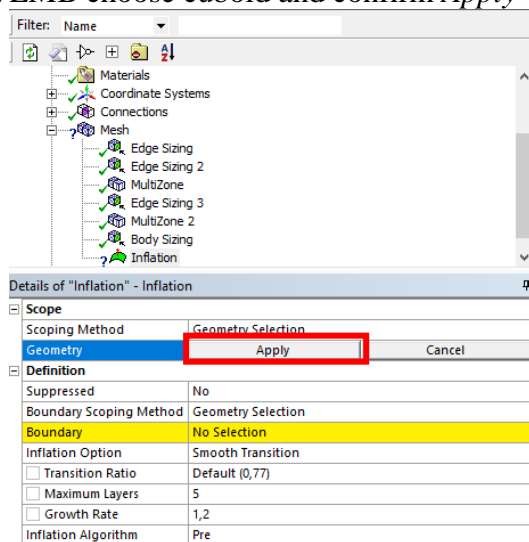
16) RMB in the tree on *Geometry* of the pipe and select *Hide Body*



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



18) LMB choose cuboid and confirm *Apply*

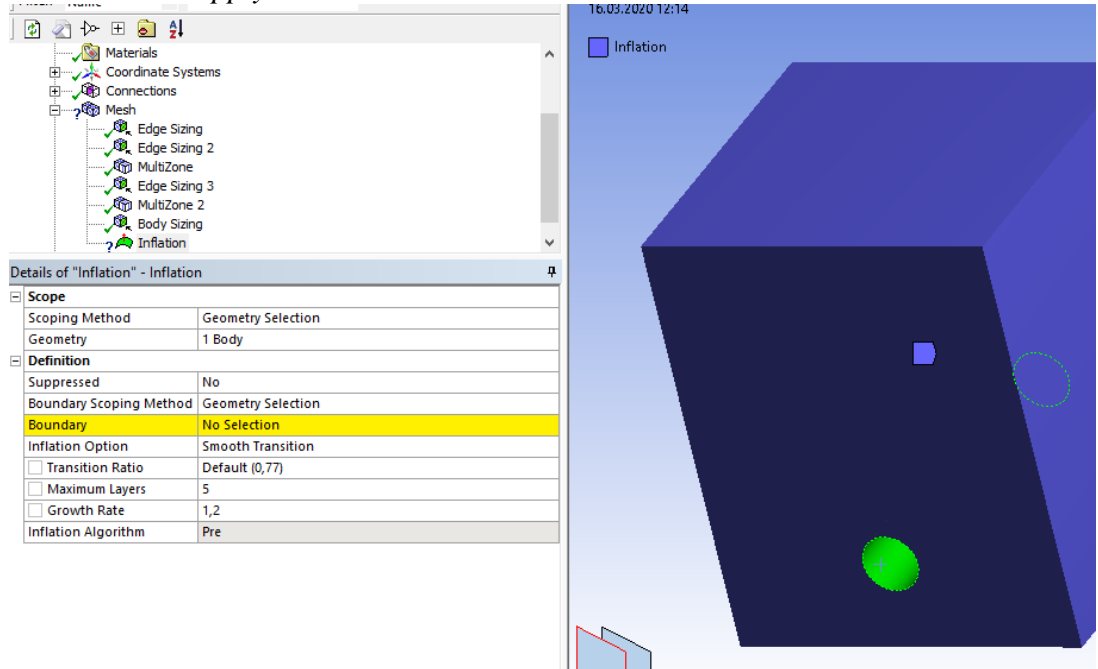


19) Change filter into face selection

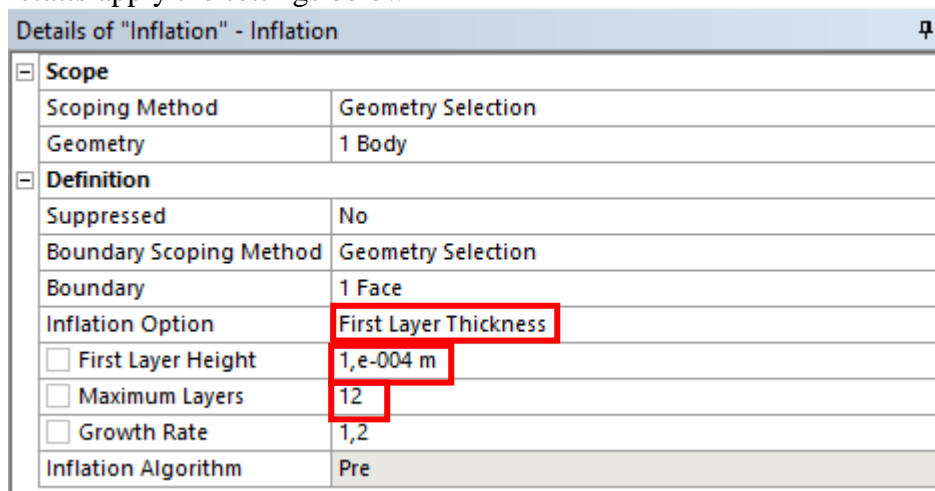




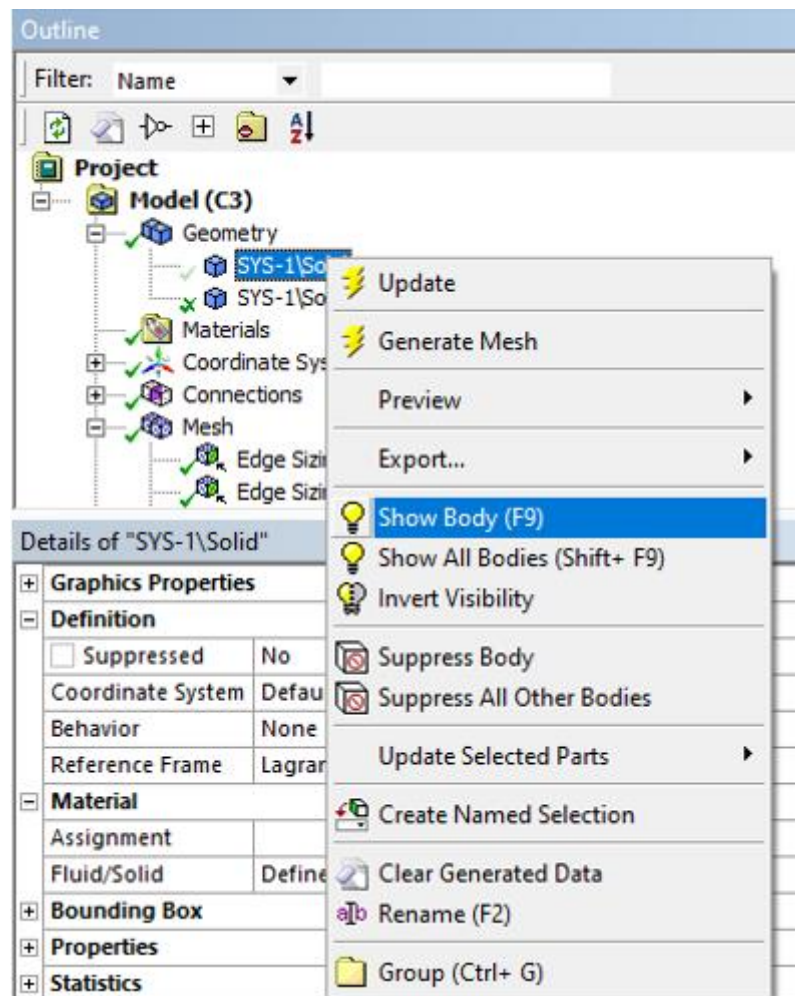
20) LMB indicate the cylindrical surface in the cuboid and LMB click in the yellow field and then *Apply*



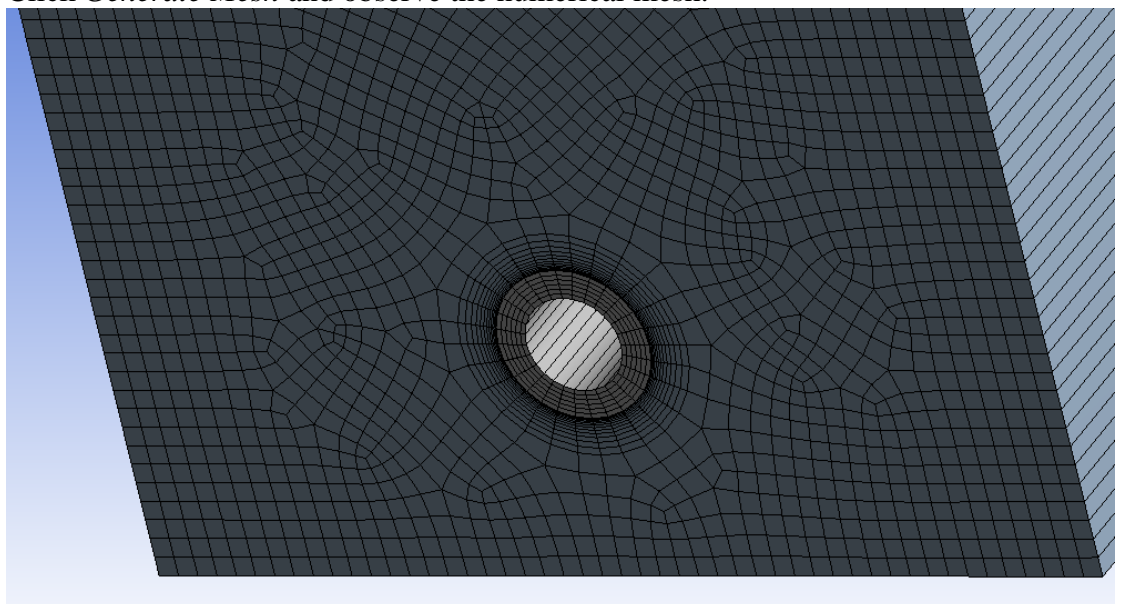
21) In *Details* apply the settings below



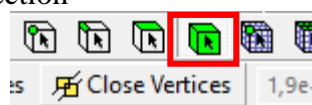
22) Turn on the visibility of the pipe



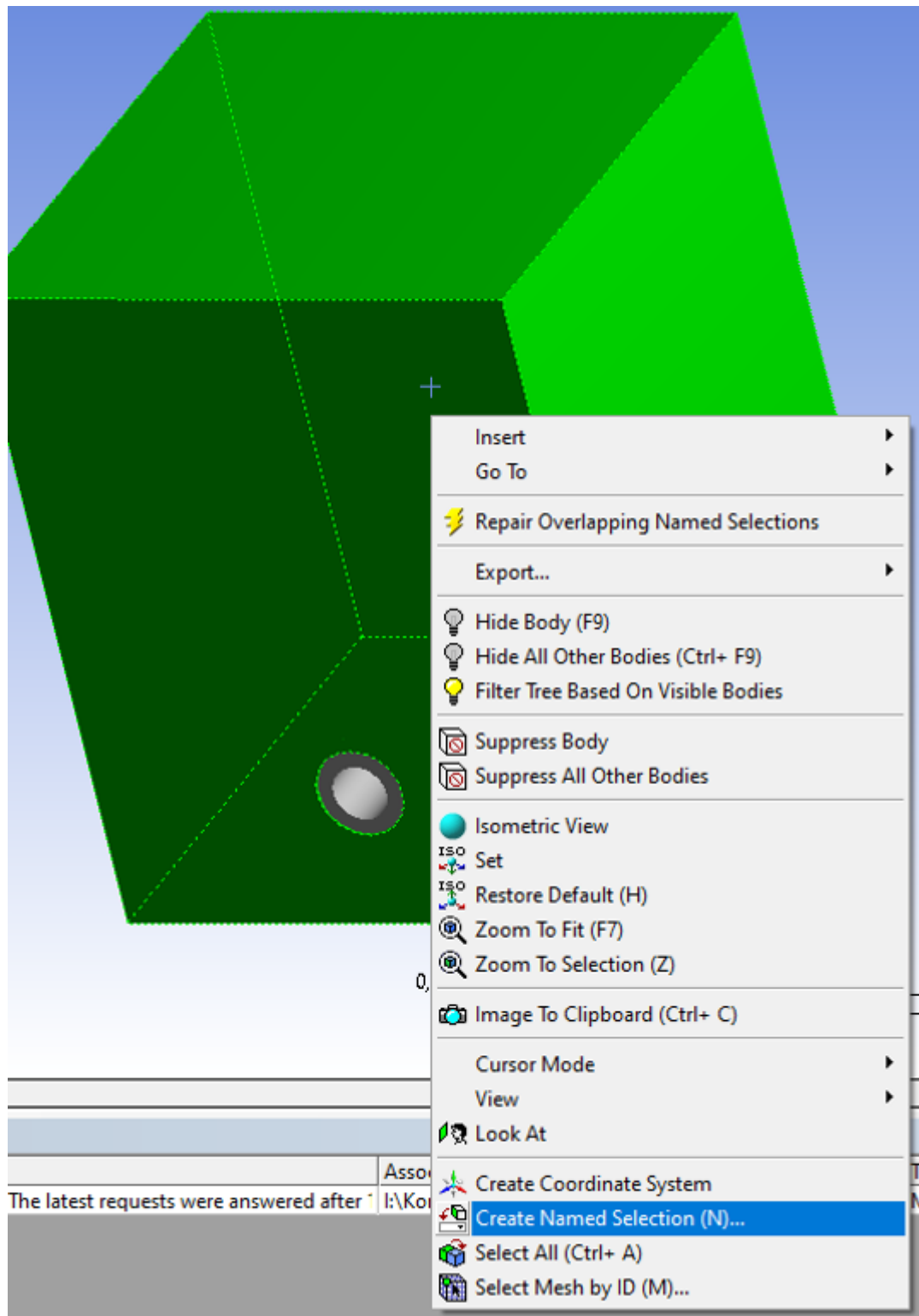
23) Click *Generate Mesh* and observe the numerical mesh.



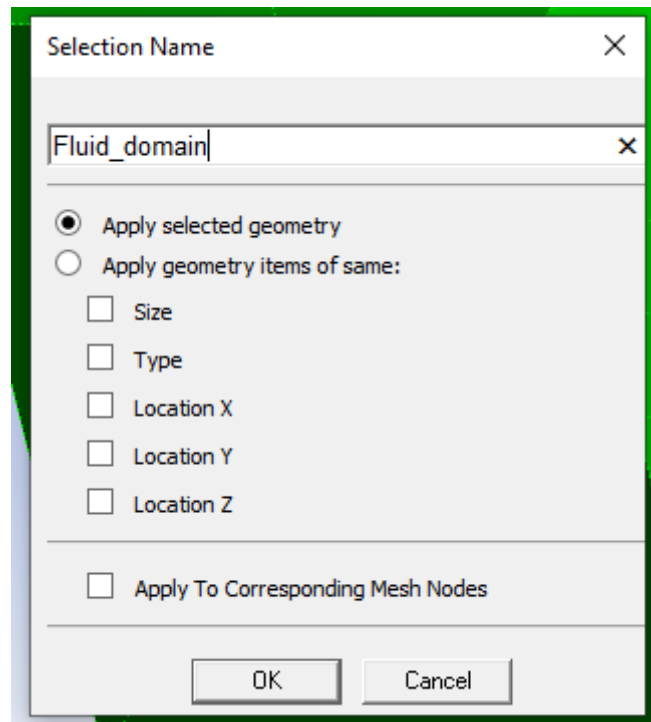
24) Change filter into body selection



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



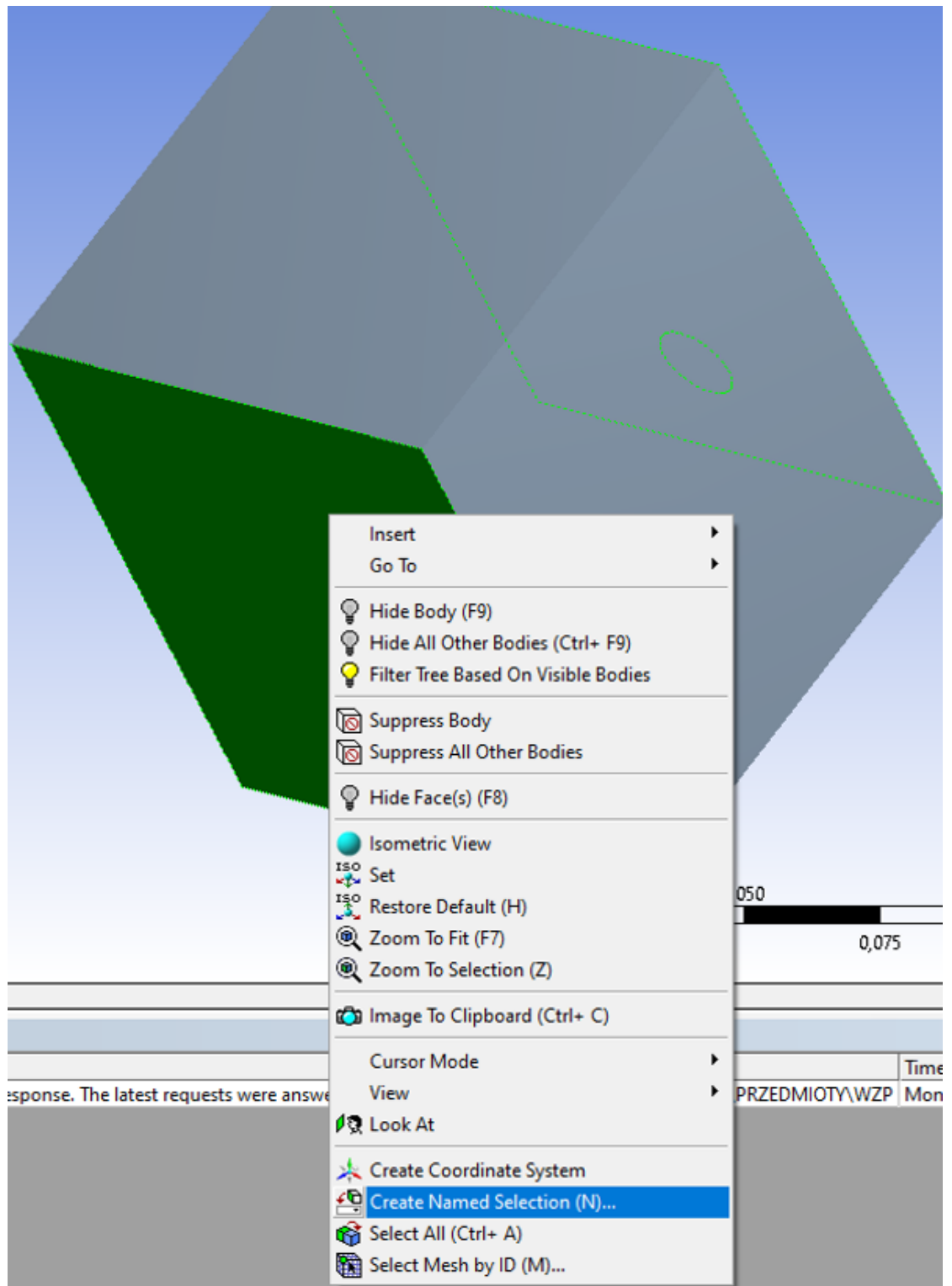
26) Name the body *Fluid\_domain*



27) Change filter into face selection

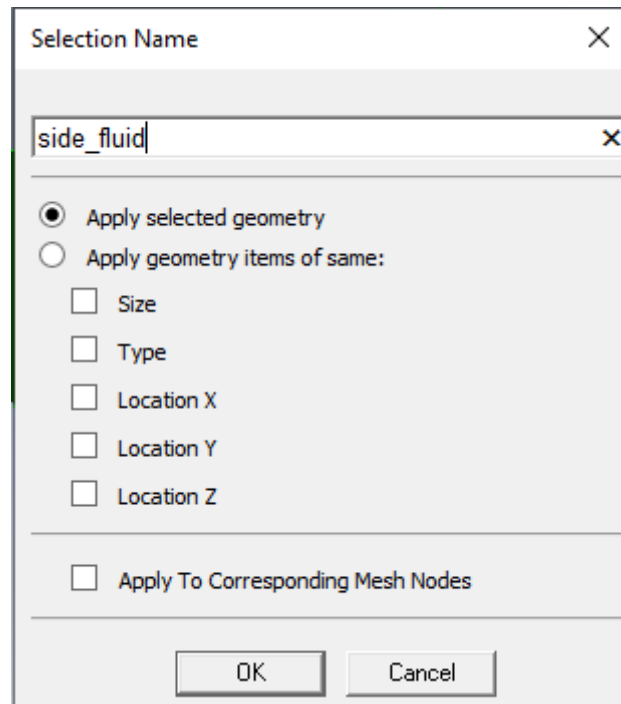


28) With *Ctrl* pressed choose two frontal faces in the cuboid



29) Name it *side\_fluid*





A screenshot of a 'Selection Name' dialog box. The title bar says 'Selection Name' with a close button (X) on the right. Below the title bar is a text input field containing 'side\_fluid' and a small 'x' icon on the right. Below the text field are two radio buttons: 'Apply selected geometry' (which is selected) and 'Apply geometry items of same:'. Under the second radio button are five checkboxes: 'Size', 'Type', 'Location X', 'Location Y', and 'Location Z'. Below these is a checkbox labeled 'Apply To Corresponding Mesh Nodes'. At the bottom are 'OK' and 'Cancel' buttons.

Selection Name

side\_fluid

☒ Apply selected geometry

☐ Apply geometry items of same:

☐ Size

☐ Type

☐ Location X

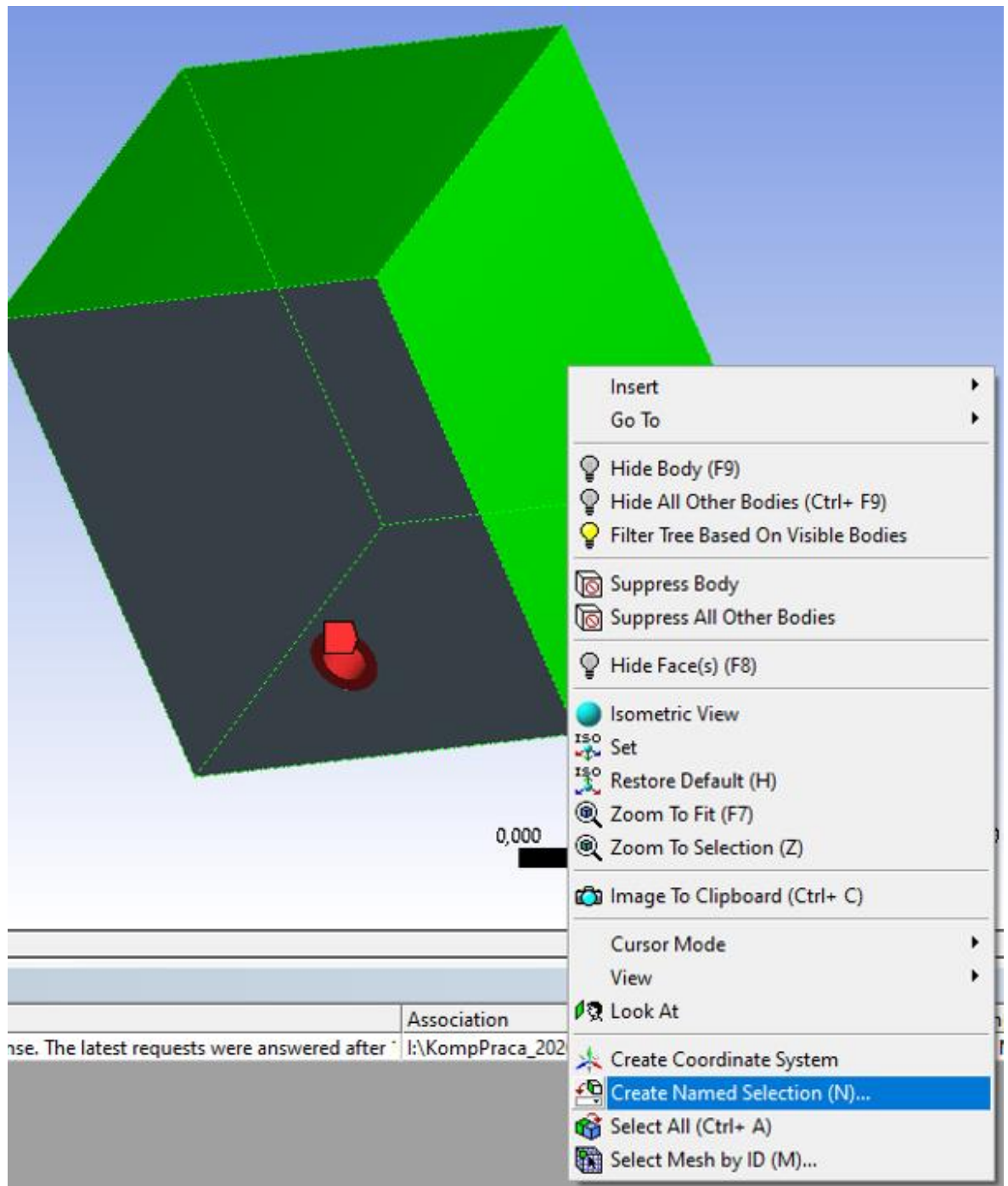
☐ Location Y

☐ Location Z

☐ Apply To Corresponding Mesh Nodes

OK Cancel

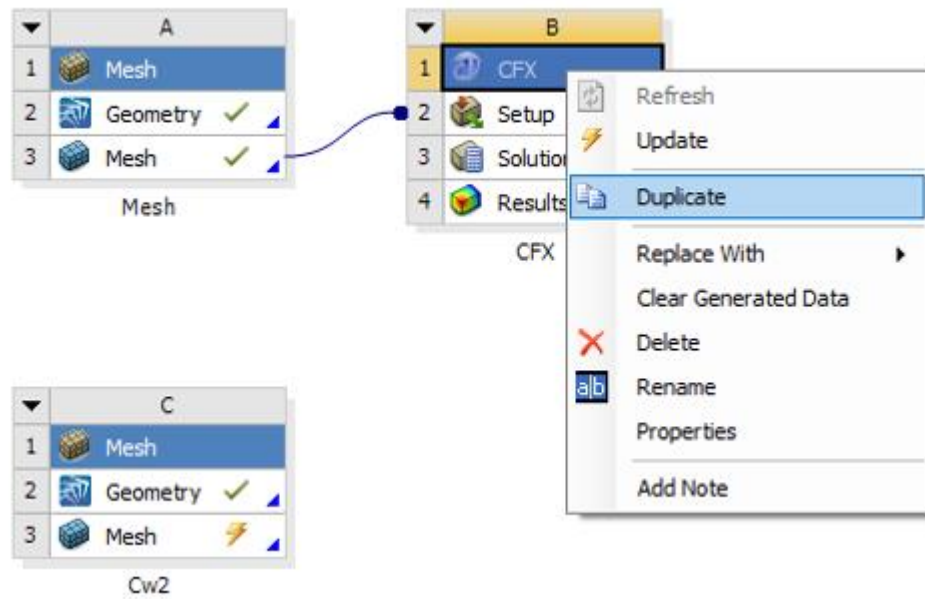
30) Similarly, select the remaining four walls of the cuboid and name them *walls*



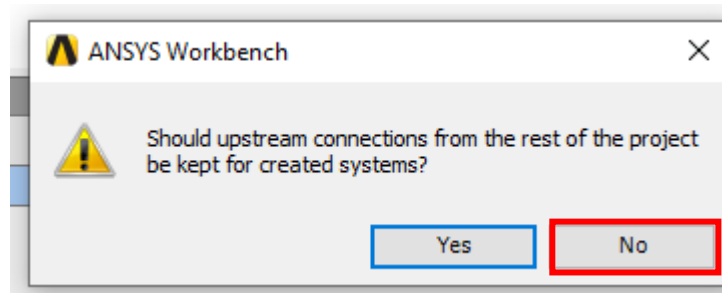
31) Close *Ansys Meshing* and save project in *Workbench*.

## 2.3. SYMULACJA NUMERYCZNA

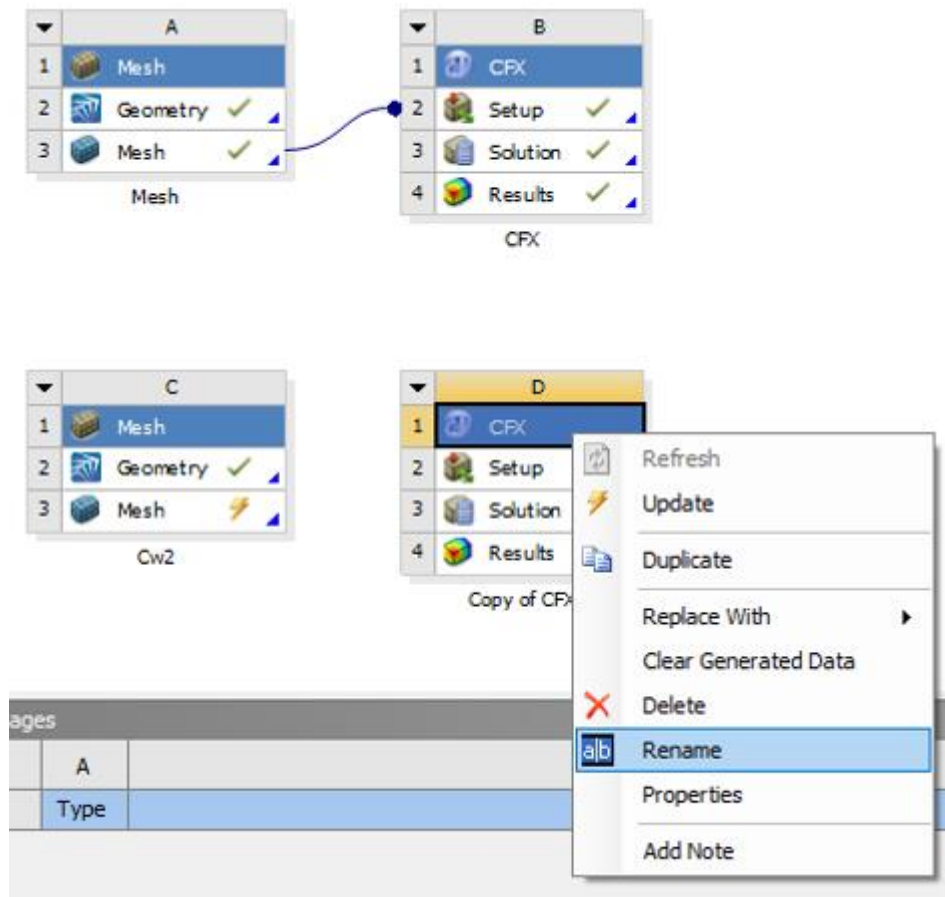
1) RMB on *CFX* from *Exercise no. 1* and select *Duplicate*



2) Then select *No*

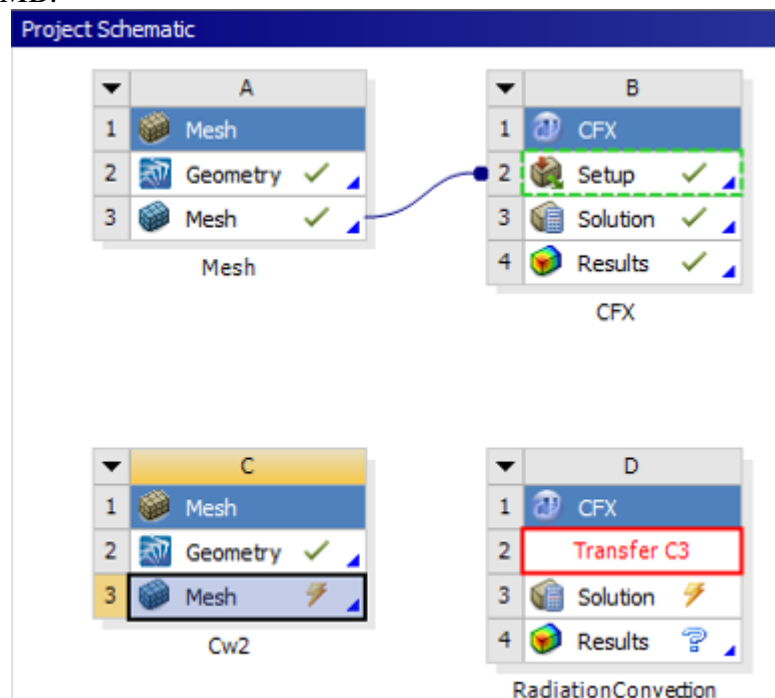


3) Change name into *RadiationConvection*

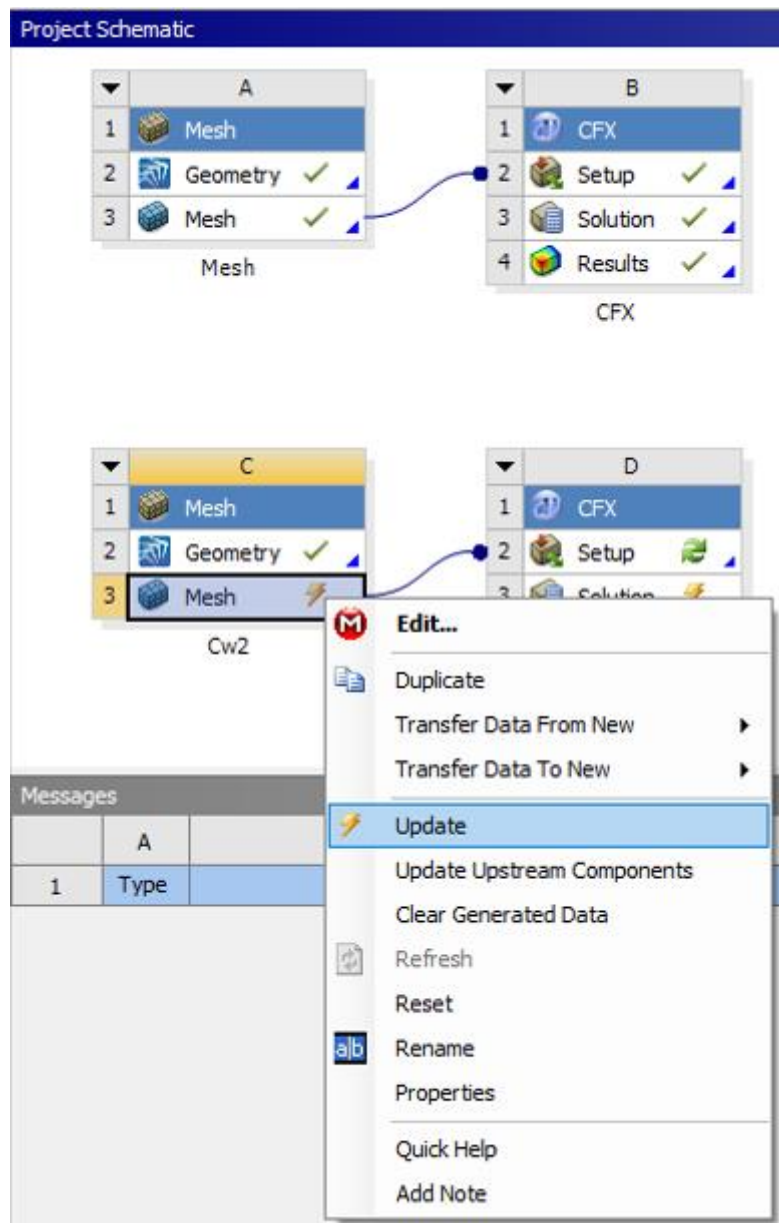


**Note:** if you don't see cell D, use Scroll to zoom out in *Project Schematic*. Then you can move cell D with LMB.

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

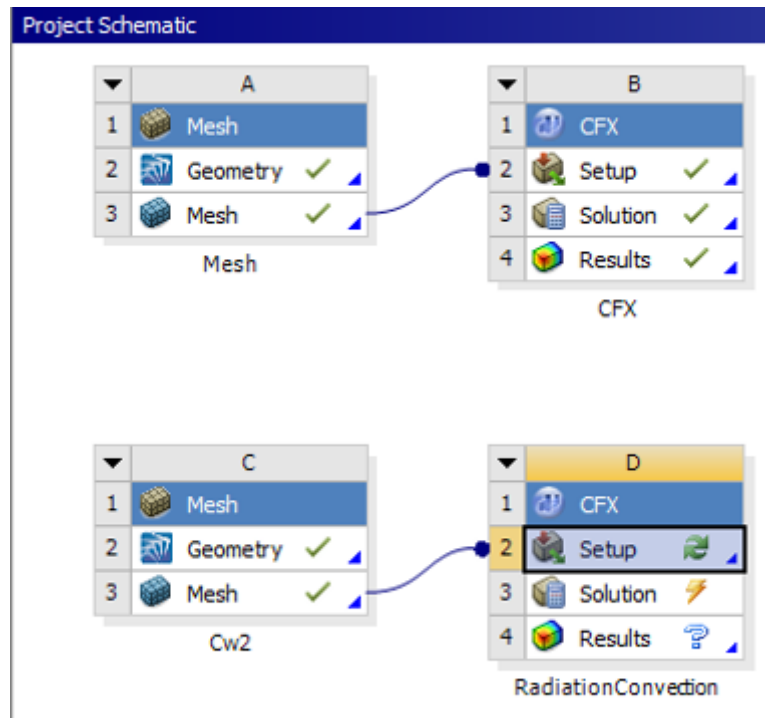


- 5) RMB on *Mesh* in *Cw2* and select *Update*

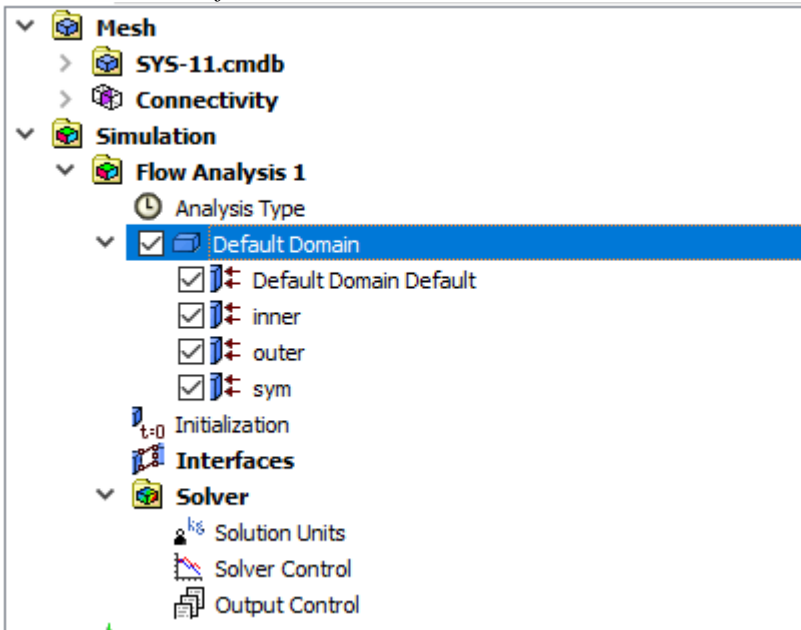


- 6) Double click LMB on *Setup* to edit a numerical model with a new numerical grid

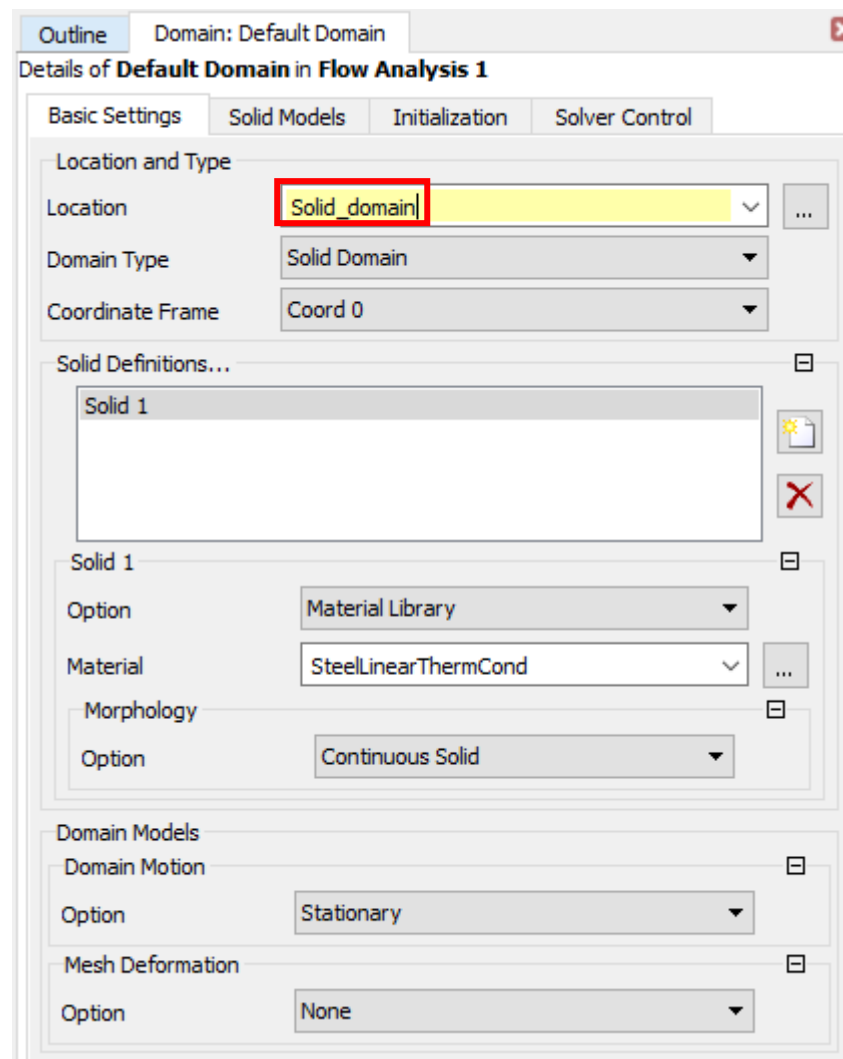




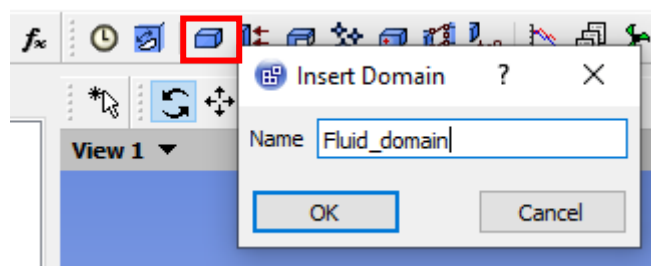
7) Double click LMB on *Default Domain*



8) Apply the settings below.



9) Create a fluid domain and name it *Fluid\_domain*



10) Apply the settings below

OutlineDomain: Fluid\_domain

Details of **Fluid\_domain** in **Flow Analysis 1**

Basic SettingsFluid ModelsInitialization

Location and Type

LocationFluid\_domain

Domain TypeFluid Domain

Coordinate FrameCoord 0

Fluid and Particle Definitions...

Fluid 1

Fluid 1

OptionMaterial Library

MaterialAir at 25 C

Morphology

OptionContinuous Fluid

☐ Minimum Volume Fraction

Domain Models

Pressure

Reference Pressure1 [atm]

Buoyancy Model

OptionBuoyant

Gravity X Dirn.0 [m s<sup>-2</sup>]

Gravity Y Dirn.-9.81 [m s<sup>-2</sup>]

Gravity Z Dirn.0 [m s<sup>-2</sup>]

Buoy. Ref. Temp.20 [C]

Ref. Location

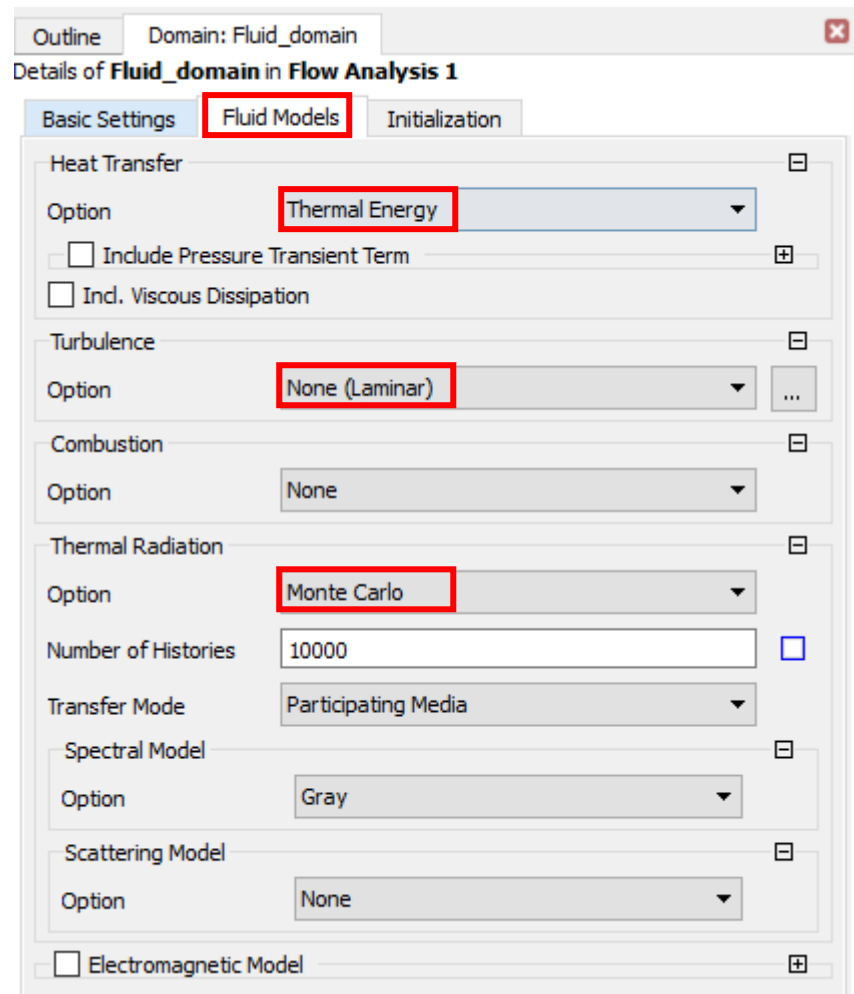
OptionAutomatic

Domain Motion

OptionStationary

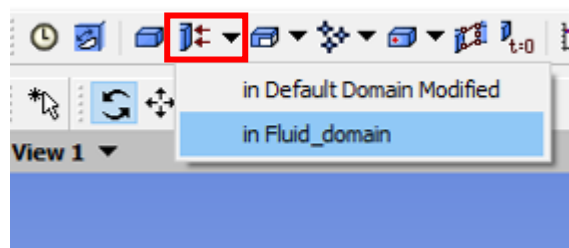
Mesh Deformation

OptionNone

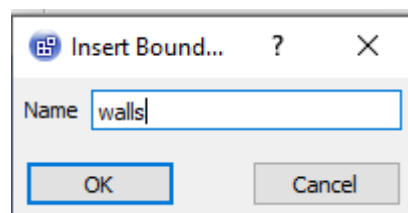


11) Confirm *OK*.

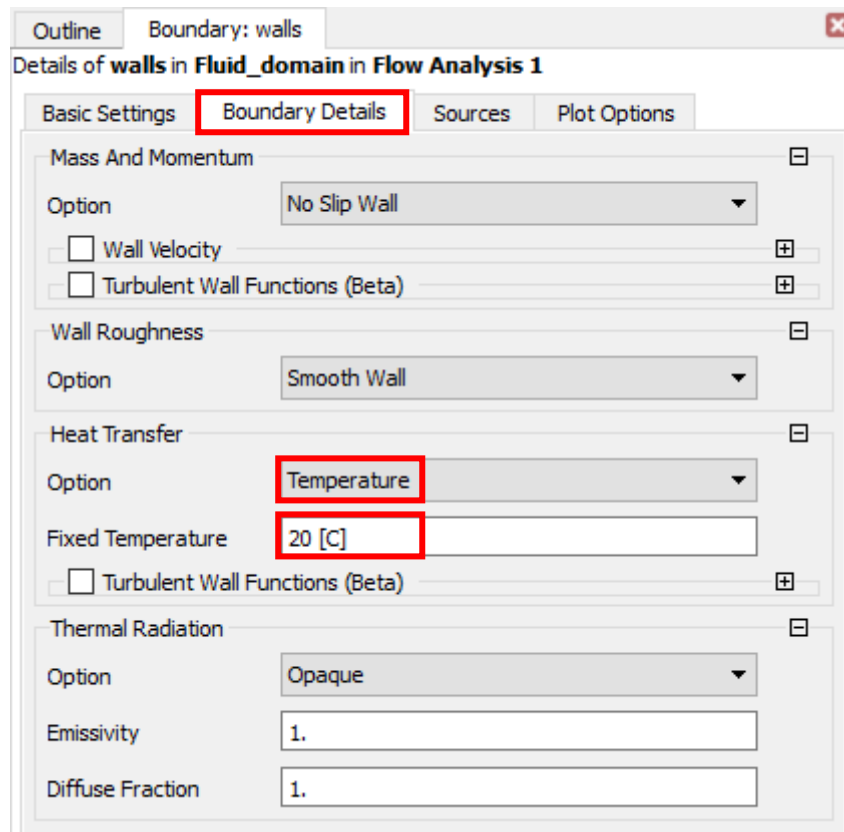
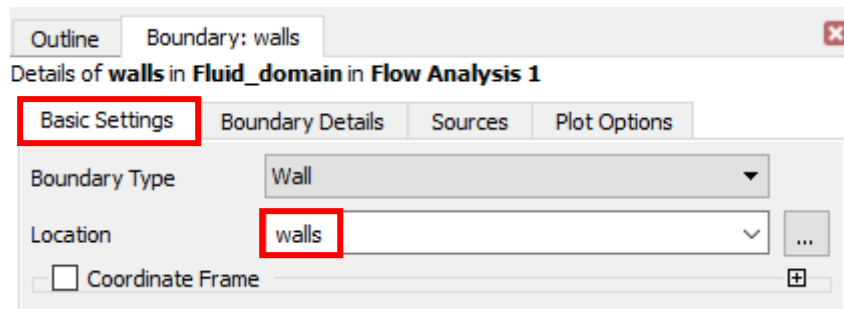
12) Create a boundary condition in the fluid domain



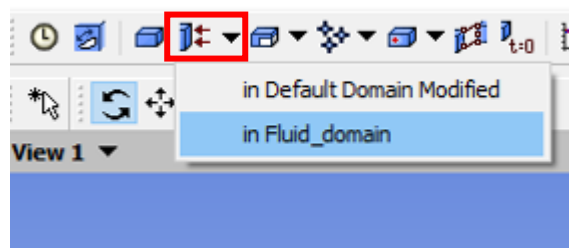
13) Name it *walls*



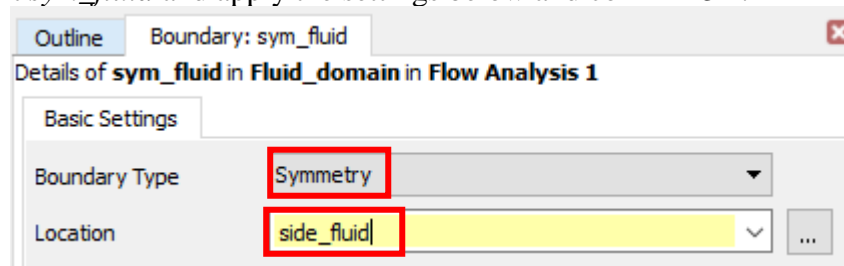
14) Make the following settings and confirm *OK*.



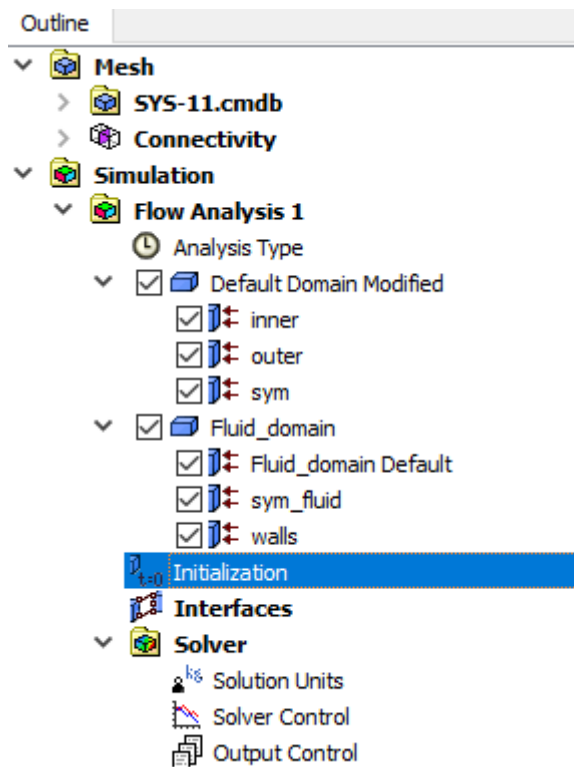
15) Create a boundary condition in the fluid domain




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*.


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 [m s<sup>-1</sup>] 


V 0 [m s<sup>-1</sup>] 


W 0 [m s<sup>-1</sup>] 


Static Pressure 


Option Automatic with Value 


Relative Pressure 0 [Pa] 


Temperature 


Option Automatic with Value 

Temperature 20 [C] 

Turbulence 

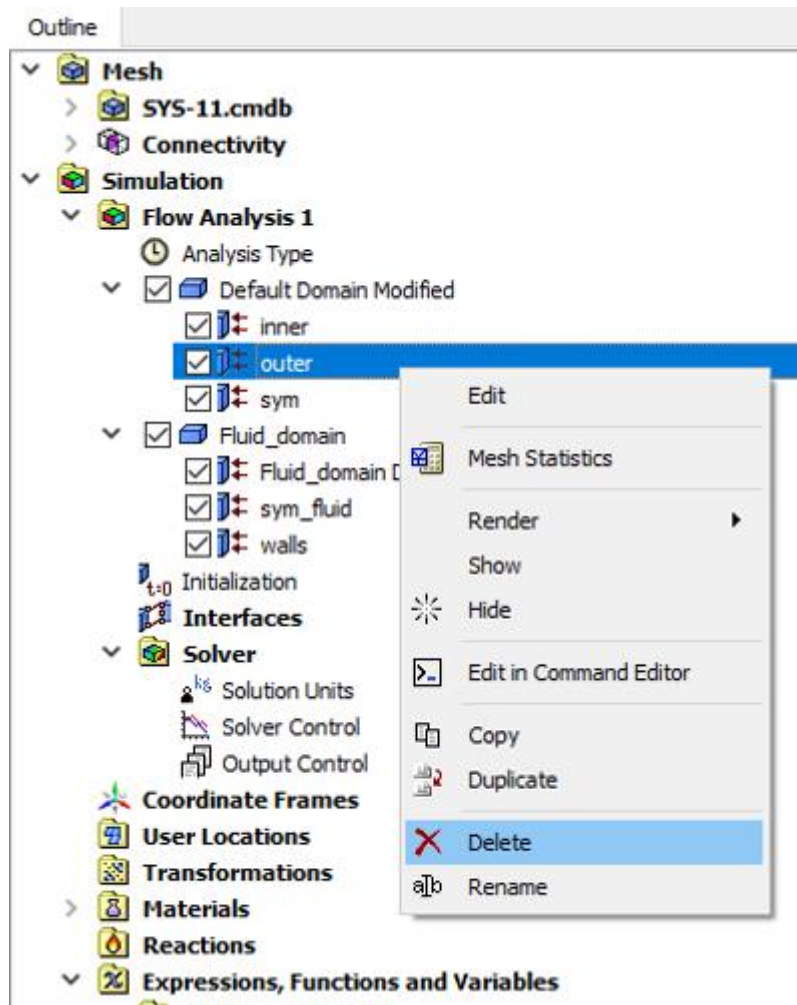
Option Medium (Intensity = 5%) 

Radiation Intensity 

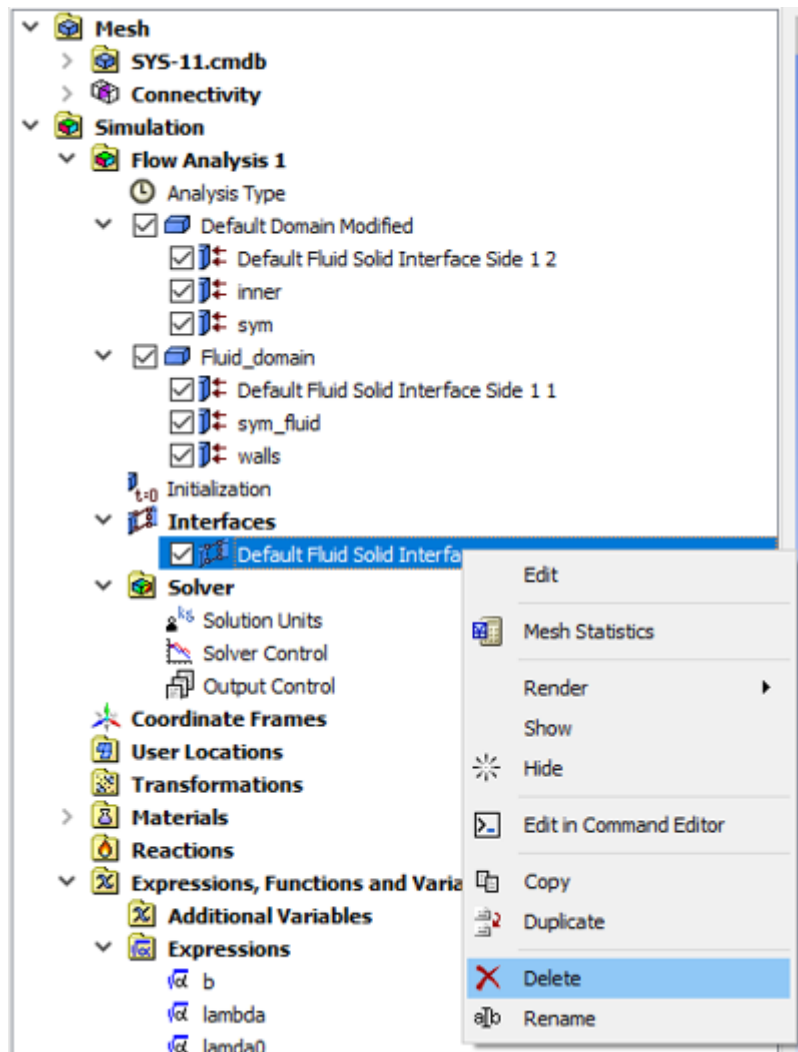
Option Automatic 

19) Delete *outer* boundary condition

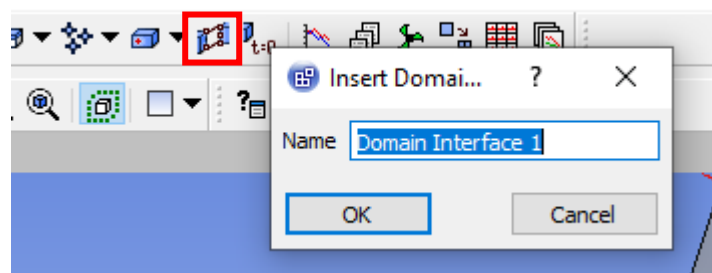




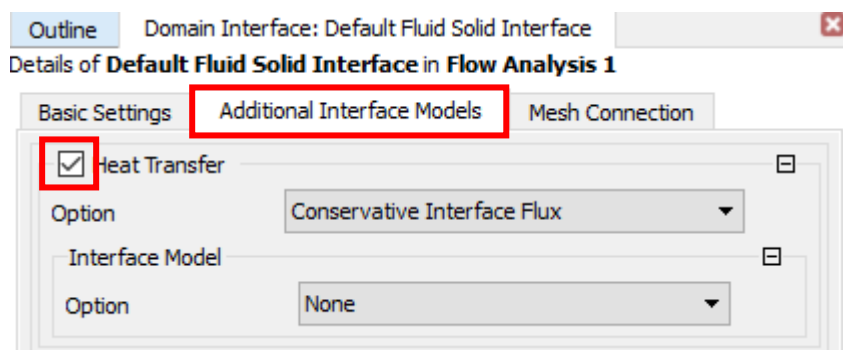
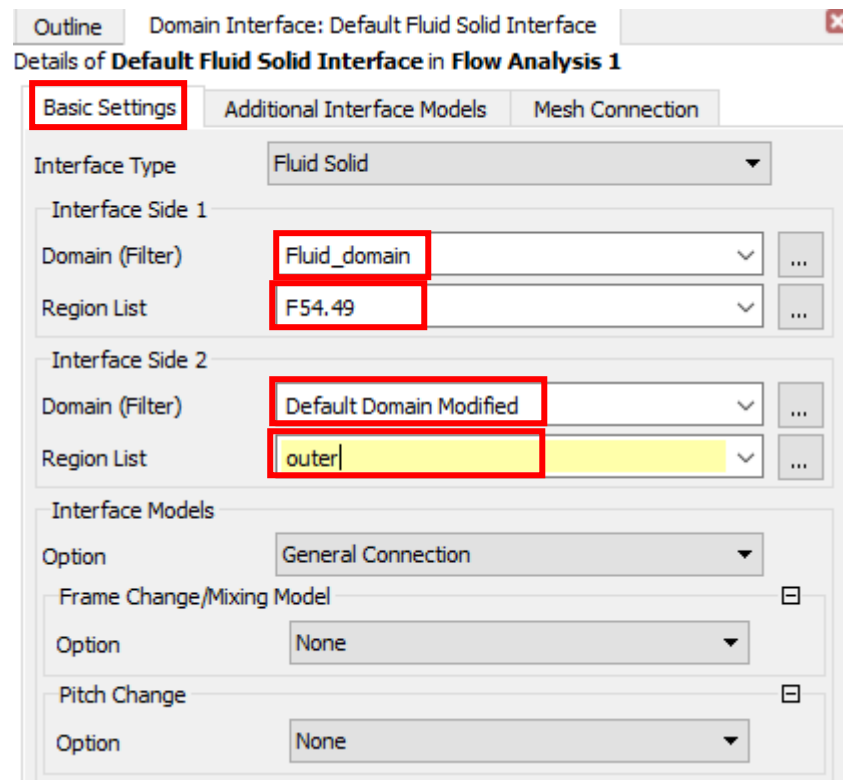
20) Delete the default settings for *Interface*



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*.

**Project Schematic**

The Project Schematic displays four components arranged in a 2x2 grid:

- Component A:** Contains a Mesh (1), Geometry (2), and Mesh (3). The label "Mesh" is centered below the list.
- Component B:** Contains CFX (1), Setup (2), Solution (3), and Results (4). The label "CFX" is centered below the list.
- Component C:** Contains a Mesh (1), Geometry (2), and Mesh (3). The label "Cw2" is centered below the list.
- Component D:** Contains CFX (1), Setup (2), Solution (3), and Results (4). The label "RadiationConvec" is centered below the list.

Connections are shown between the Mesh (3) of Component A and the Setup (2) of Component B, and between the Mesh (3) of Component C and the Setup (2) of Component D.

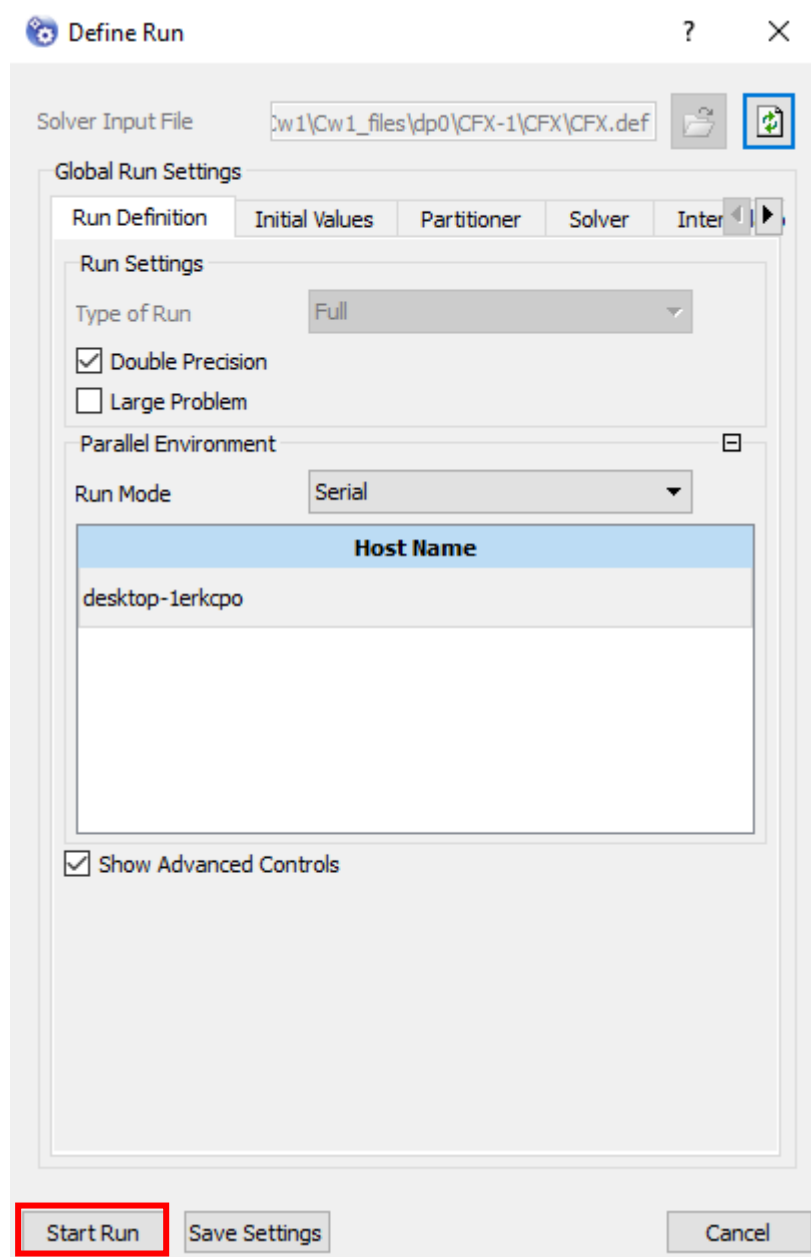
A context menu is open for the Solution (3) component in Component D. The menu options are:

- Edit...
- Display Monitors
- Duplicate
- Transfer Data From New
- Transfer Data To New
- Update
- Continue Calculation
- Update Upstream Components
- Clear Generated Data
- Refresh
- Clear Old Solution Data
- Clear Cached Solution Data
- Clear Execution Control
- Reset
- Rename
- Properties
- Quick Help
- Add Note

**Messages**

Messages		
	A	
1	Type	

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

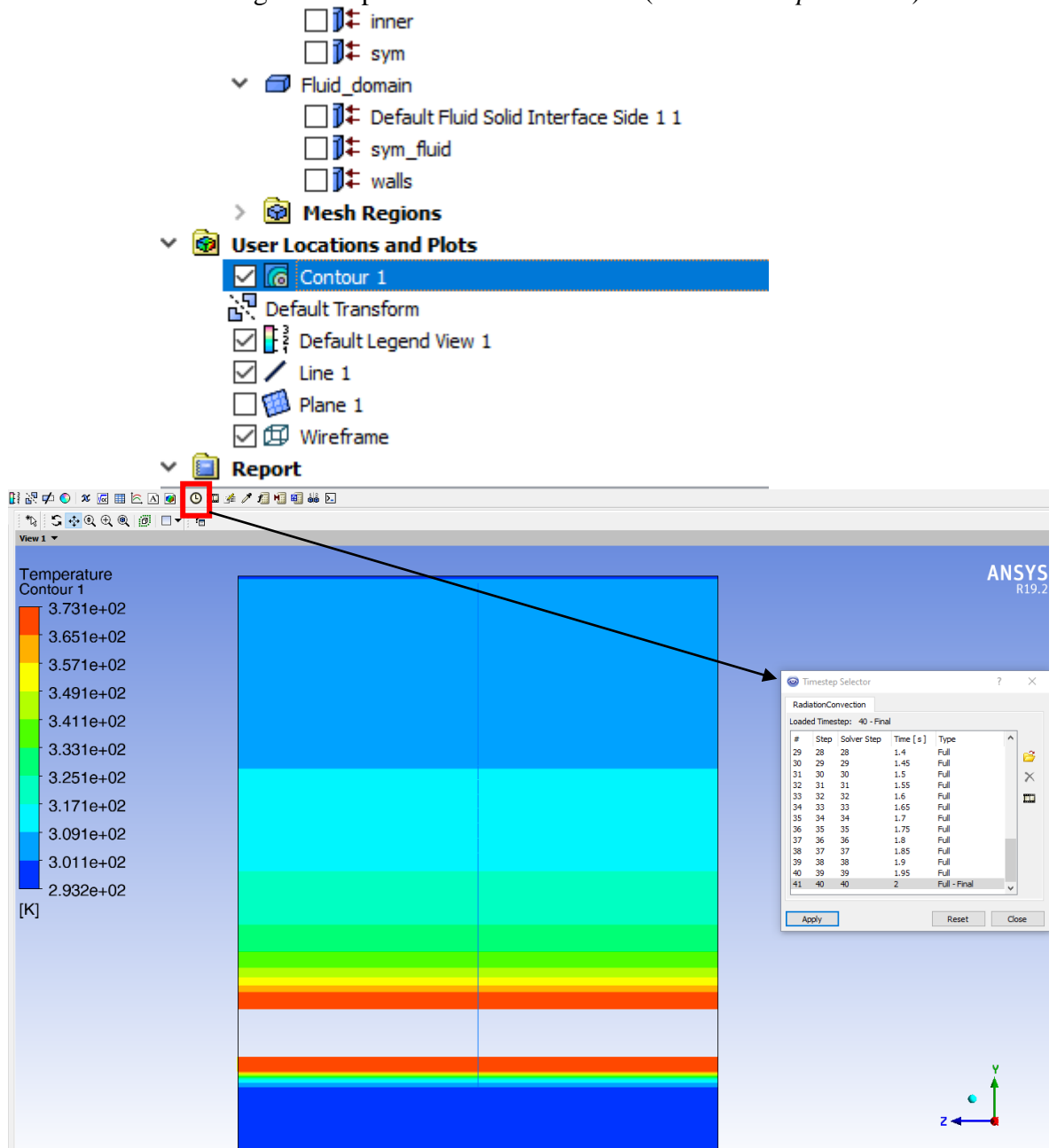
- 1) Double click LMB on *Results*.

The screenshot displays the ANSYS Workbench interface. On the left is the **Toolbox**, which is divided into **Analysis Systems** and **Component Systems**. The **CFX** component system is highlighted. The main area shows the **Project Schematic** with four cells: A (Mesh), B (CFX), C (Cw2), and D (RadiationConvection). Cell A contains a Mesh component. Cell B contains CFX components: CFX, Setup, Solution, and Results. Cell C contains a Mesh component. Cell D contains CFX components: CFX, Setup, Solution, and Results. A blue line connects the Mesh component in Cell A to the Setup component in Cell B. Another blue line connects the Mesh component in Cell C to the Setup component in Cell D. The **Messages** panel at the bottom shows a table with the following content:

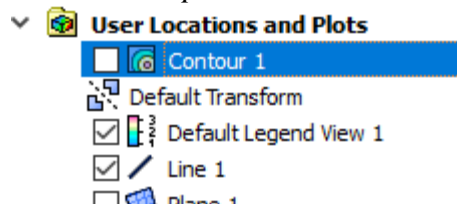
	A	
1	Type	

At the bottom of the interface, there is a status bar that reads "Starting CFD-Post..."

- 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*)

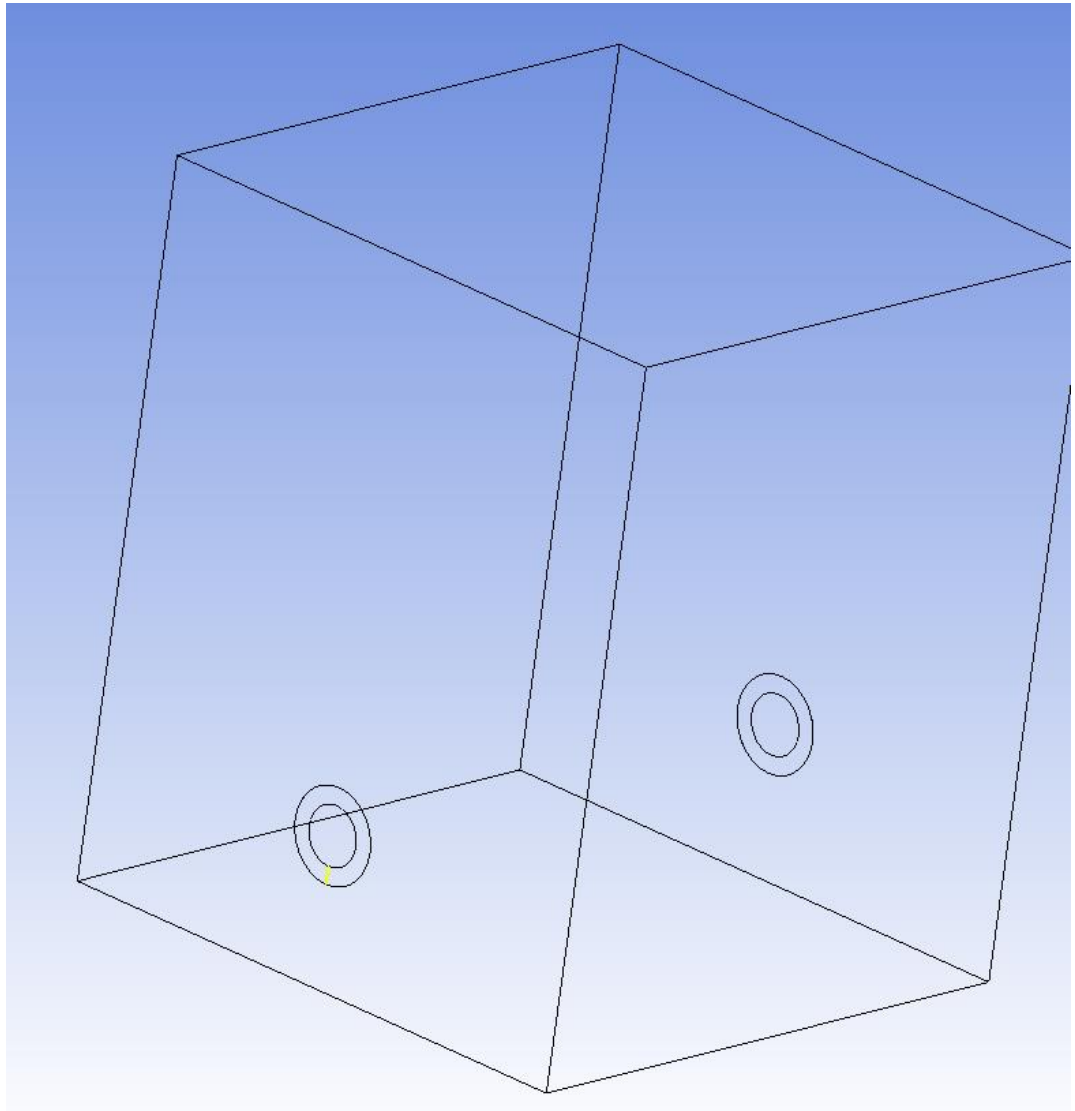


- 4) Turn off contours and close *Timestep Selector*

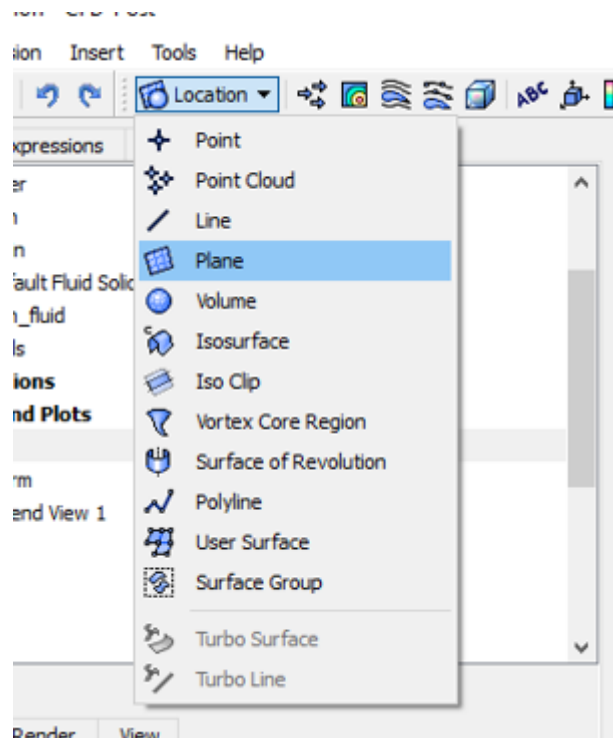


- 5) Rotate the model to view as below

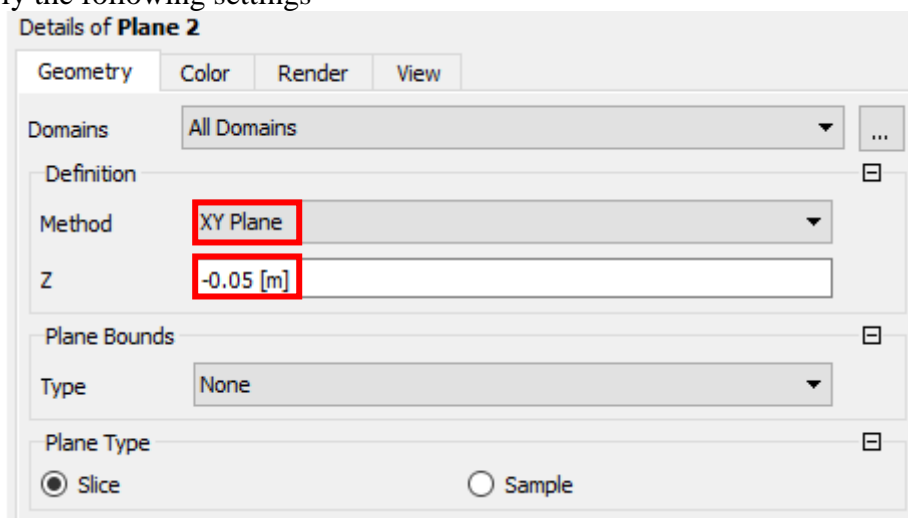




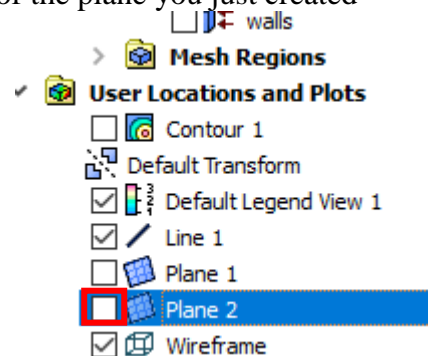
6) Create a transverse plane with *Location->Plane*



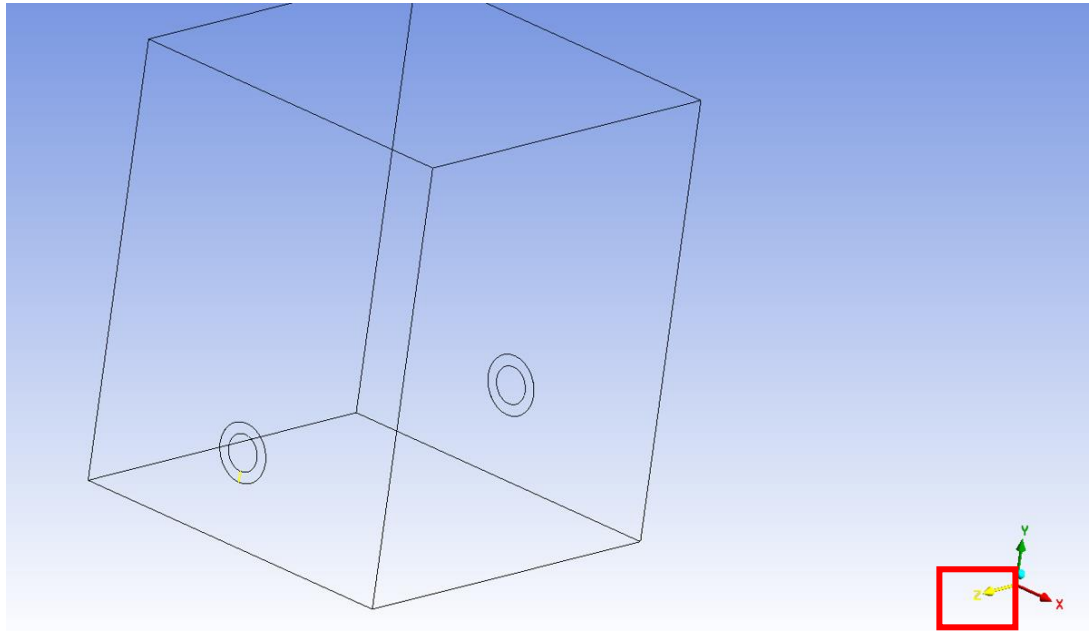
7) Apply the following settings



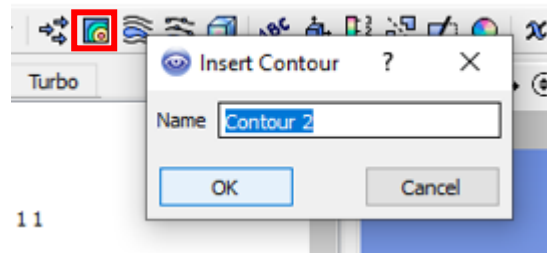
8) Turn off the visibility of the plane you just created



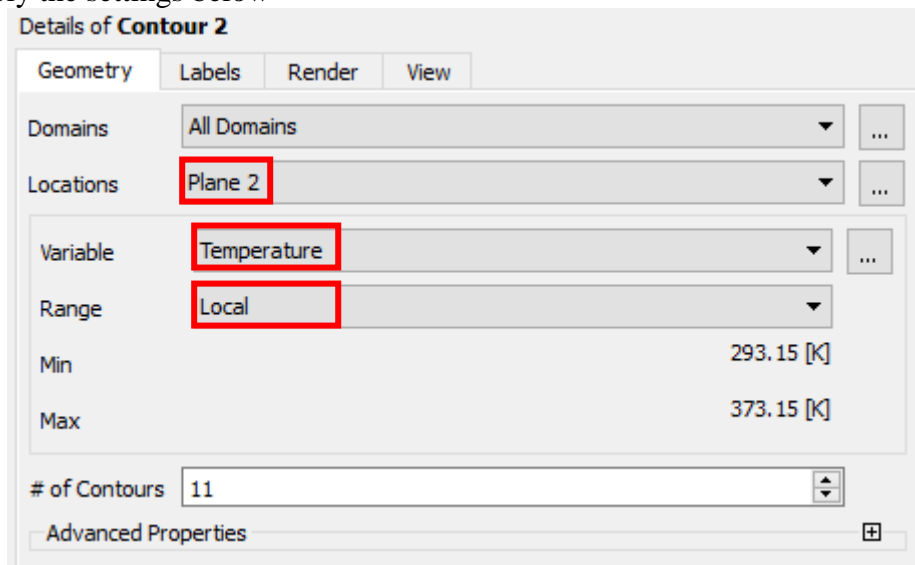
9) LMB press the Z axis



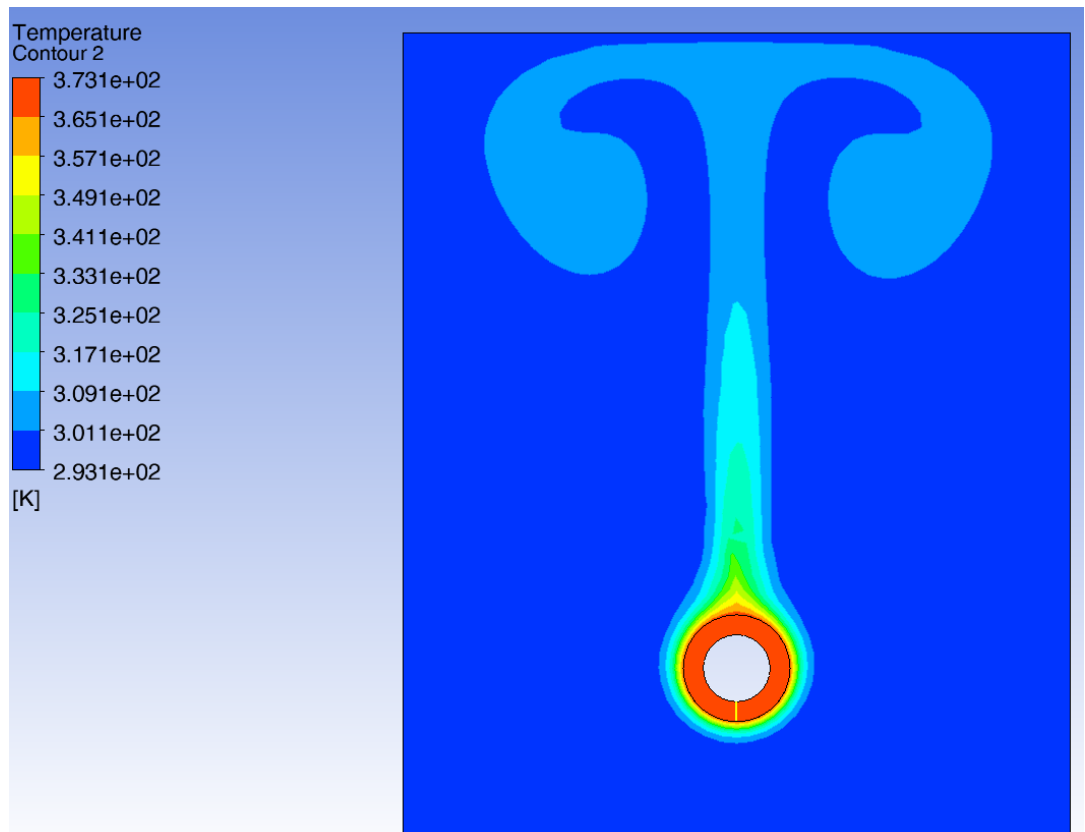
10) Create new contours



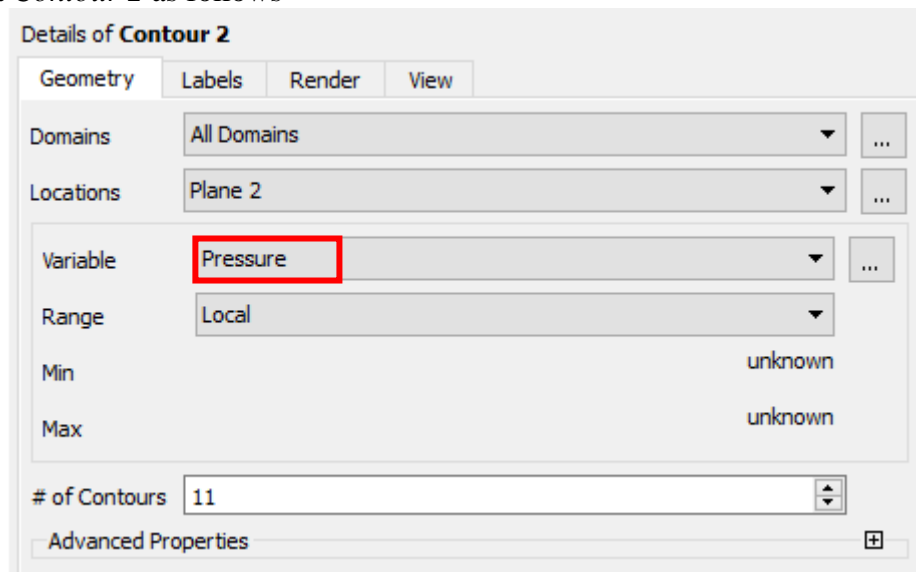
11) Apply the settings below



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



13) Edit *Contour 2* as follows



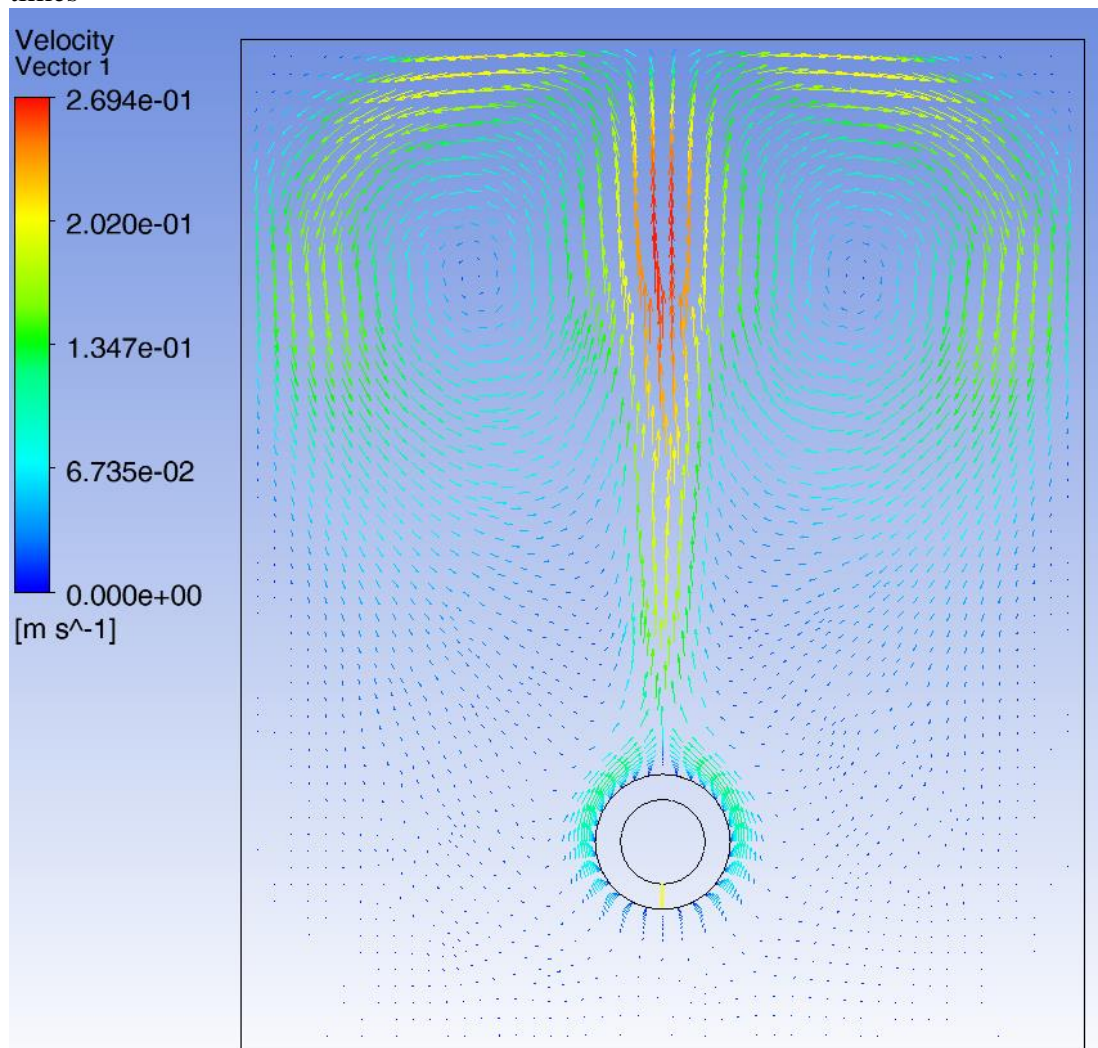
14) Using *Timestep Selector* observe pressure distributions at different times



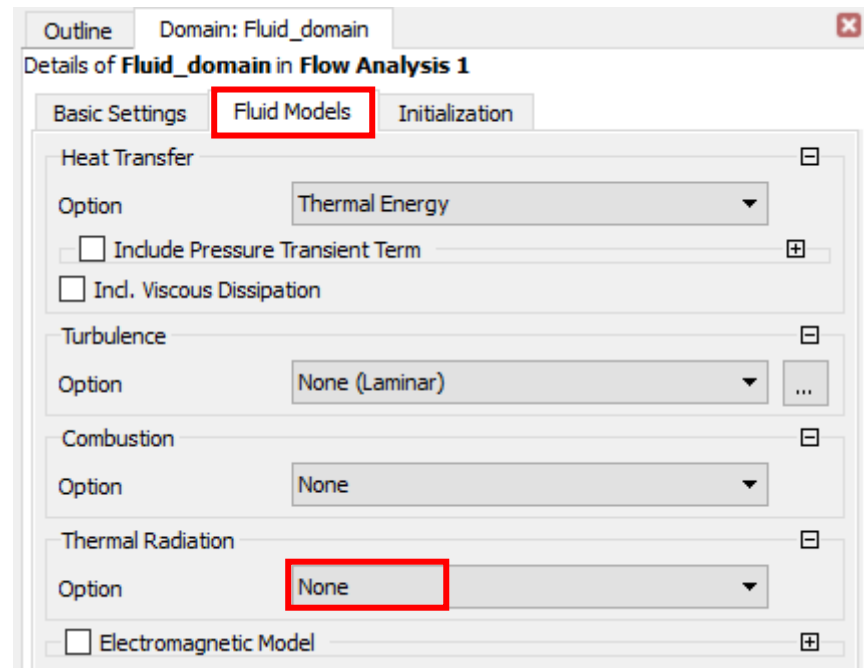
**Details of Vector 1**

Geometry	Color	Symbol	Render	View
Domains: All Domains				
Definition				
Locations	Plane 2			
Sampling	Vertex			
Reduction	Reduction Factor			
Factor	1.0			
Variable	Velocity			
Boundary Data	<input checked="" type="radio"/> Hybrid <input type="radio"/> Conservative			
Projection	None			

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:

$\text{=areaInt(Wall Heat Flux )@Domain Interface 1 Side 1}$

**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 streamlines 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.