Politechnika Wrocławska

Mechanical and Power Engineering Faculty
Full-time studies

Selected problems of thermal-flow processes

Exercise no. 1

Transient heat transfer analysis

Wrocław 2020

TABLE OF CONTENTS

1. INTRODUCTION

The exercise will show how to model the transient heat transfer in a cylindrical partition. A simulation of one-dimensional heat conduction will be carried out in a steel pipe. An additional assumption will be the linear dependence of the thermal conductivity coefficient of the pipe material on the temperature. The diagram of the analyzed case is presented in Fig. 1.

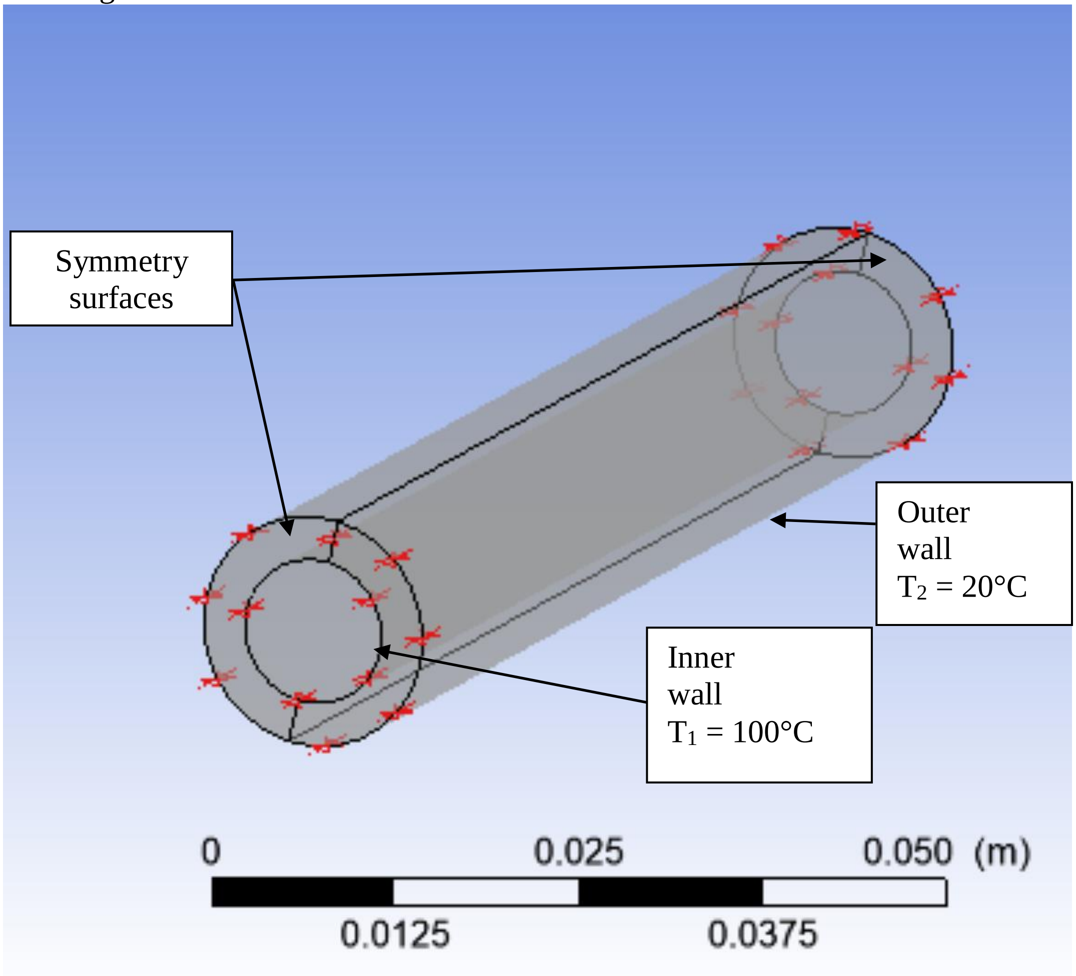


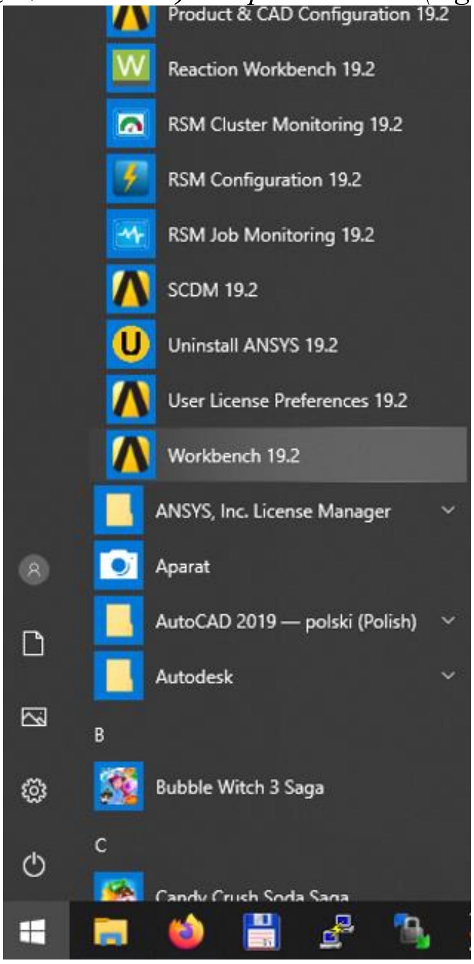
Fig. 1. Scheme of the problem of transient heat conduction in a cylindrical partition

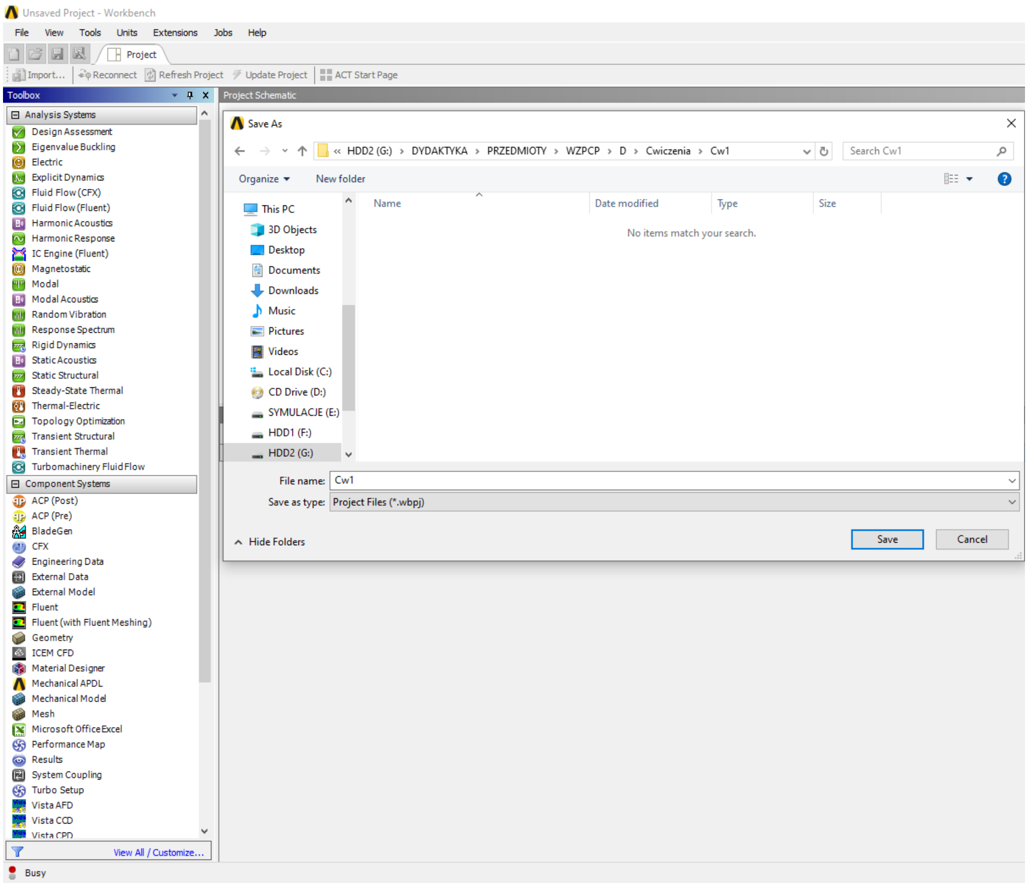
2. ONE-DIMENSIONAL TRANSIENT CONDUCTION THROUGH A CYLINDRYCAL WALL

Do the following:

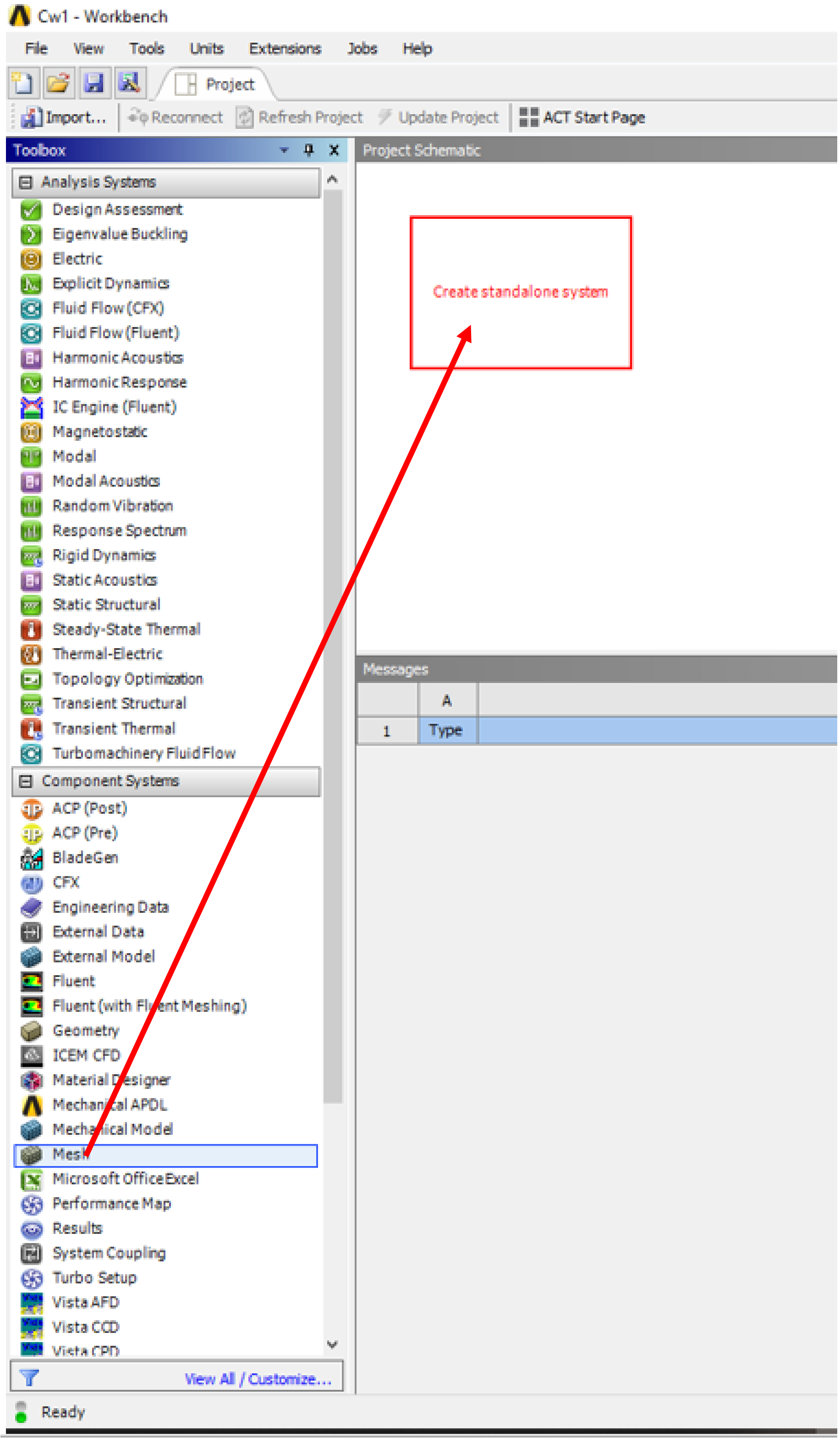
1) Open Ansys Workbench and save the project named Cw1 in the directory called Cw1 (*File->Save As*).

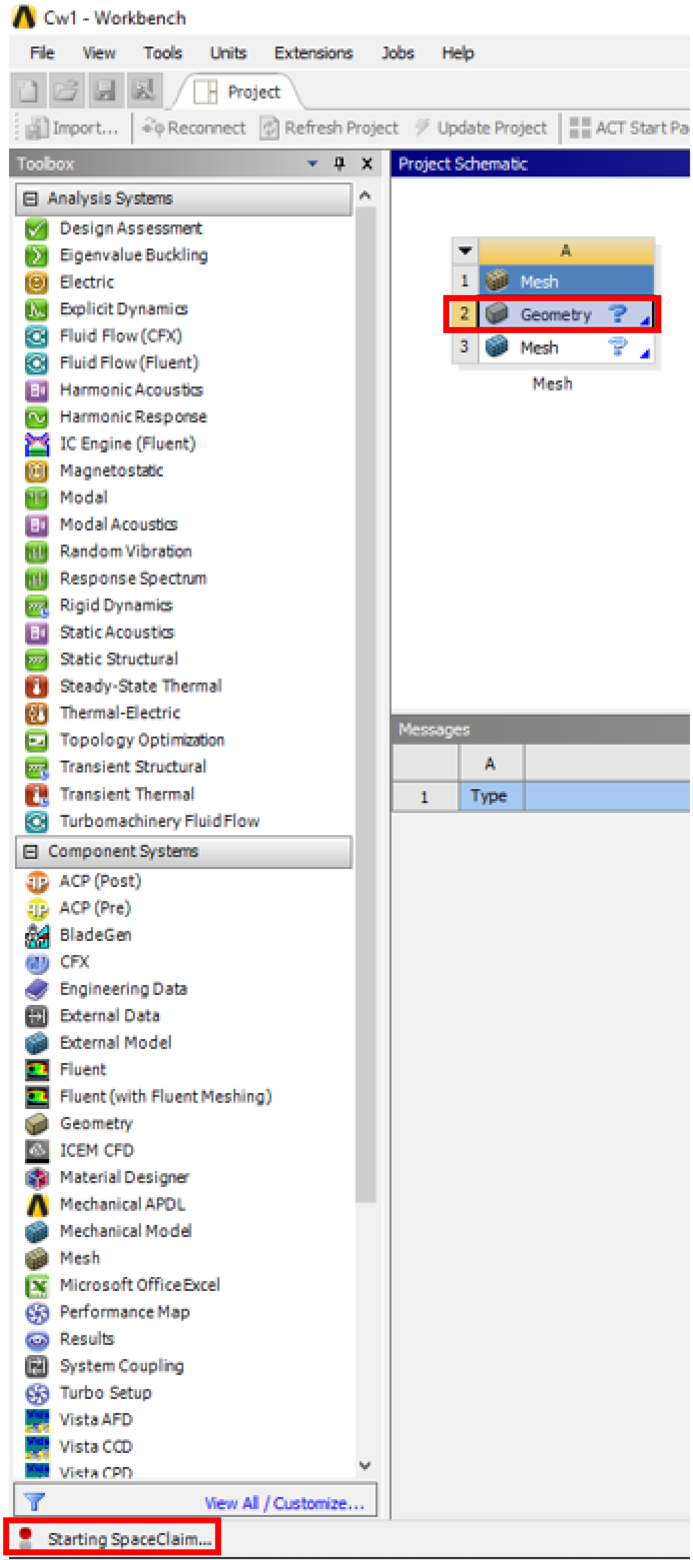
THUMB RULE NO 1: *Create a separate catalog for each project*
ZASADA PRAKTYCZNA NR 2: *In the names of catalogs do not use: space, special marks (e.g.. @#$%^&* etc.) and polish marks (e.g. ź)*





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.


Cw1 - Workbench
File
View
Tools
Units
Extensions
Jobs
Help
Project
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



3) In *Spaceclaim*, create a cylindrical pipe with the dimensions shown in Fig. 2. Set the pipe length to 100 mm.

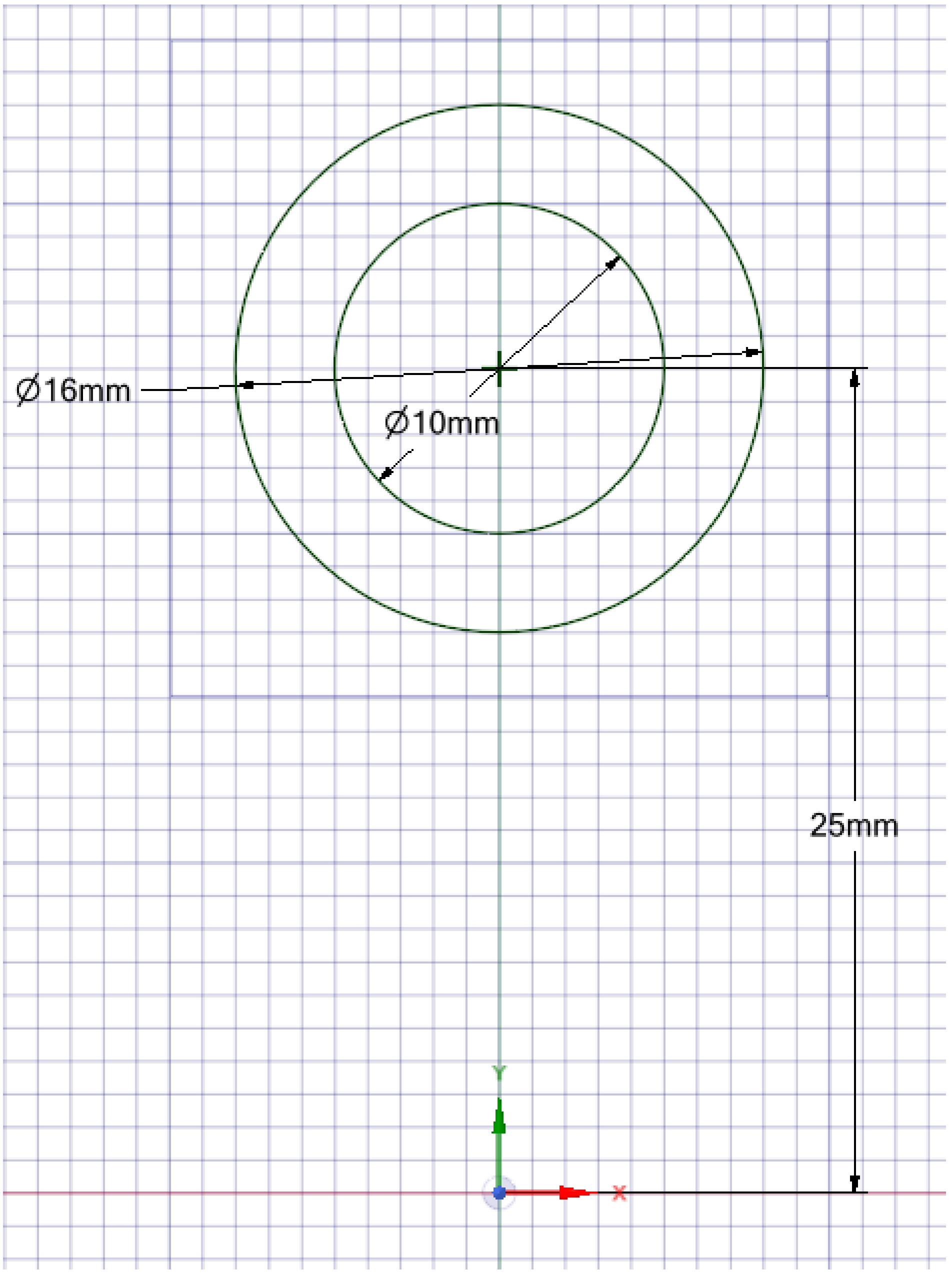
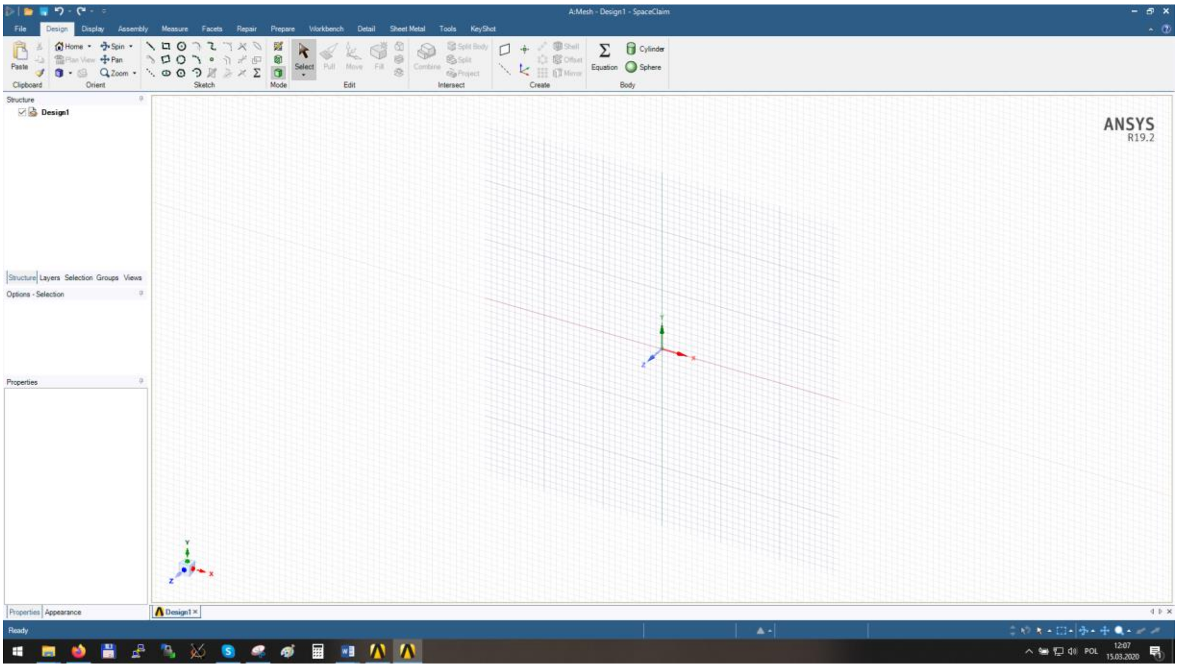
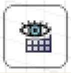


Fig. 2. Dimensions of the cylindrical pipe in cross section (pipe length is 100 mm)

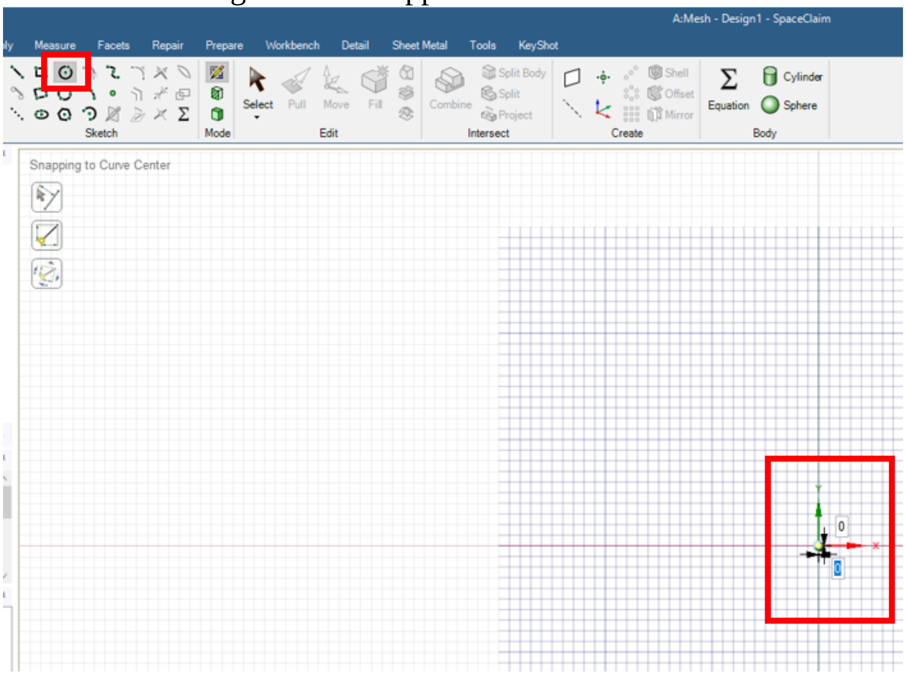
Click LMB *Select New Sketch*  to select the drawing plane.

Select the X-Y plane as shown below.

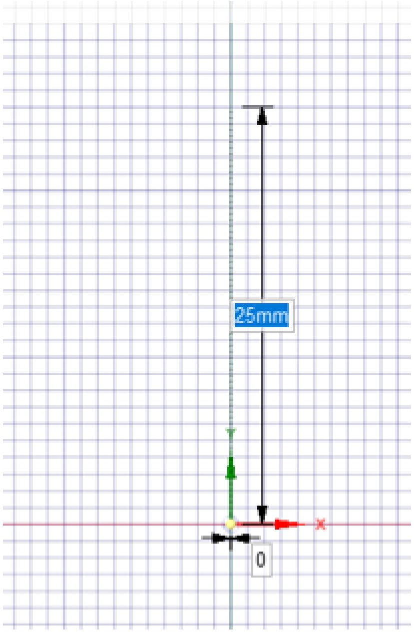


Click *Plan View*  to rotate the drawing plane parallel to the screen (you can also do this by pressing *Shift* + *v*).

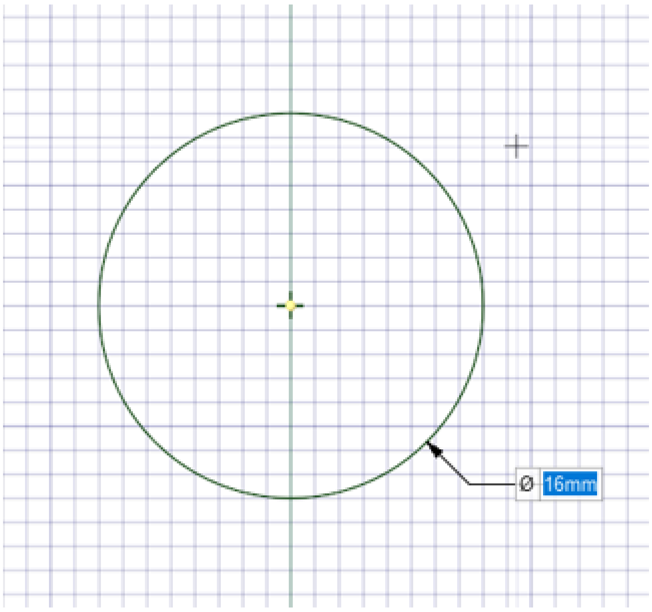
In the panel at the top of the screen select the circle drawing icon and move the cursor to the center of the coordinate system and then press the *Shift* key - two dimensions editing fields will appear as shown below.



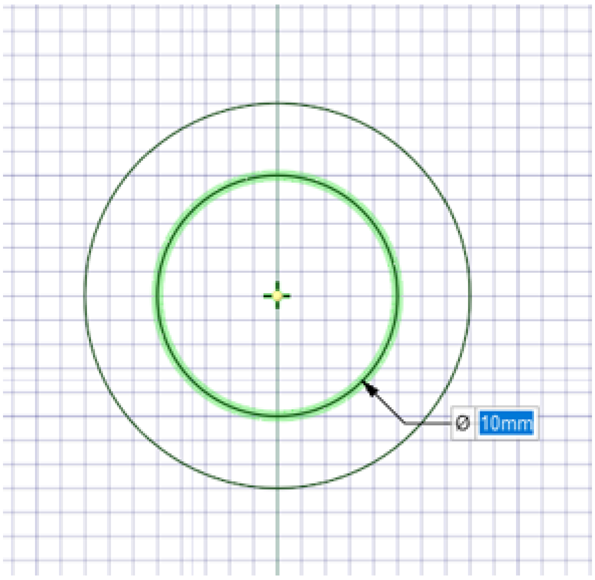
You can switch between the dimension edit fields with the *Tab* key. Set the horizontal dimension to 0 and the vertical dimension to 25 mm and press *Enter*.



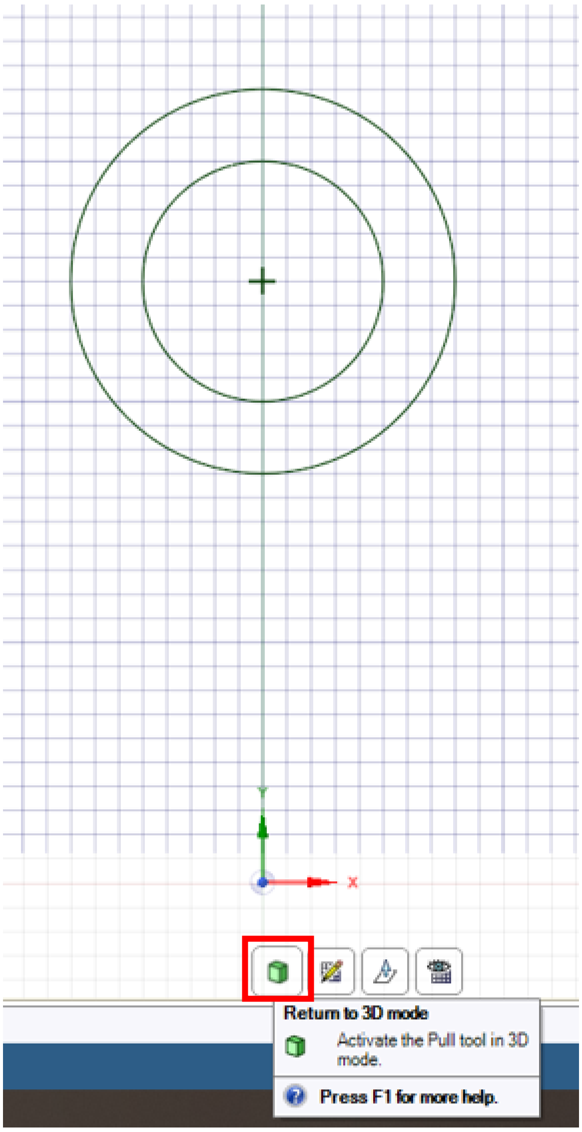
The program will proceed to drawing a circle. Enter the value 16 mm and confirm with the key *Enter*. Remember, that if you fail at any time you can click the undo icon  (located in the upper left corner of the screen) or *Ctrl + z*.



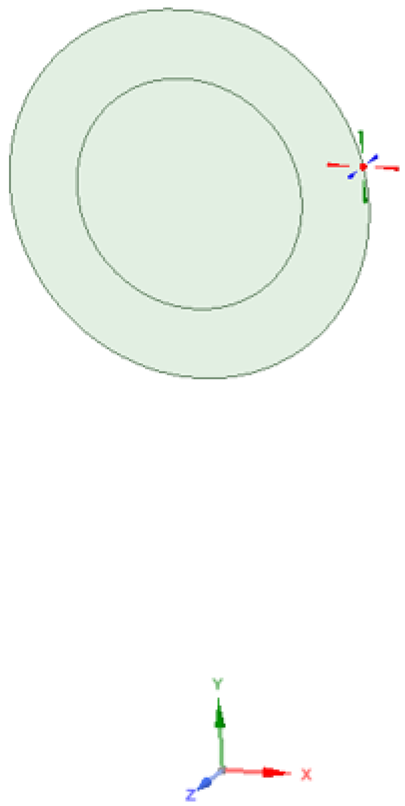
Then create a second circle 10 mm in diameter by clicking LMB in the center of the previously created circle.



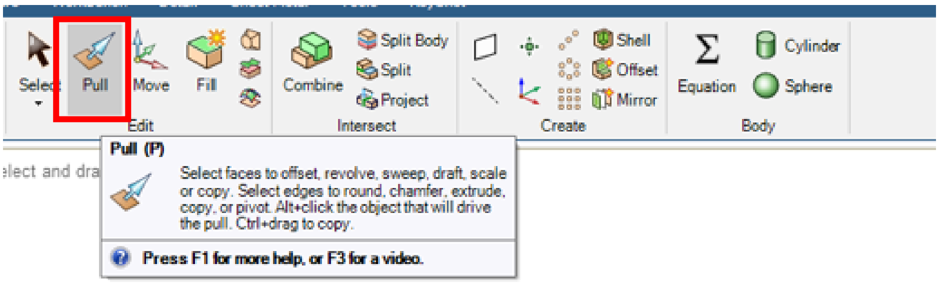
To exit the circle drawing command, press *Esc* and LMB, click the *Return to 3D mode* icon

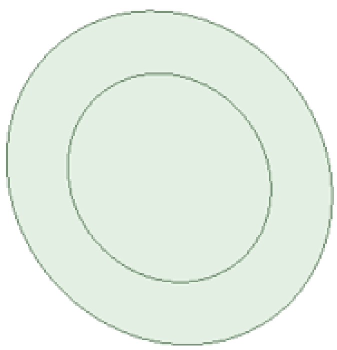


Rotate the view by holding down the *Scroll* mouse button and moving it to get an isometric view similar to the one below.

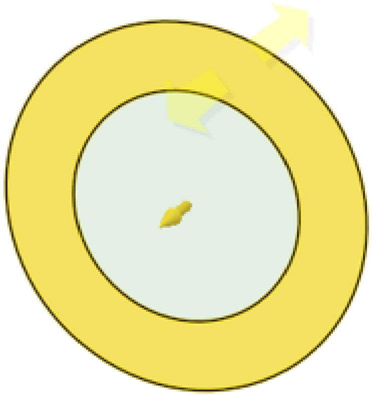


Select *Pull*

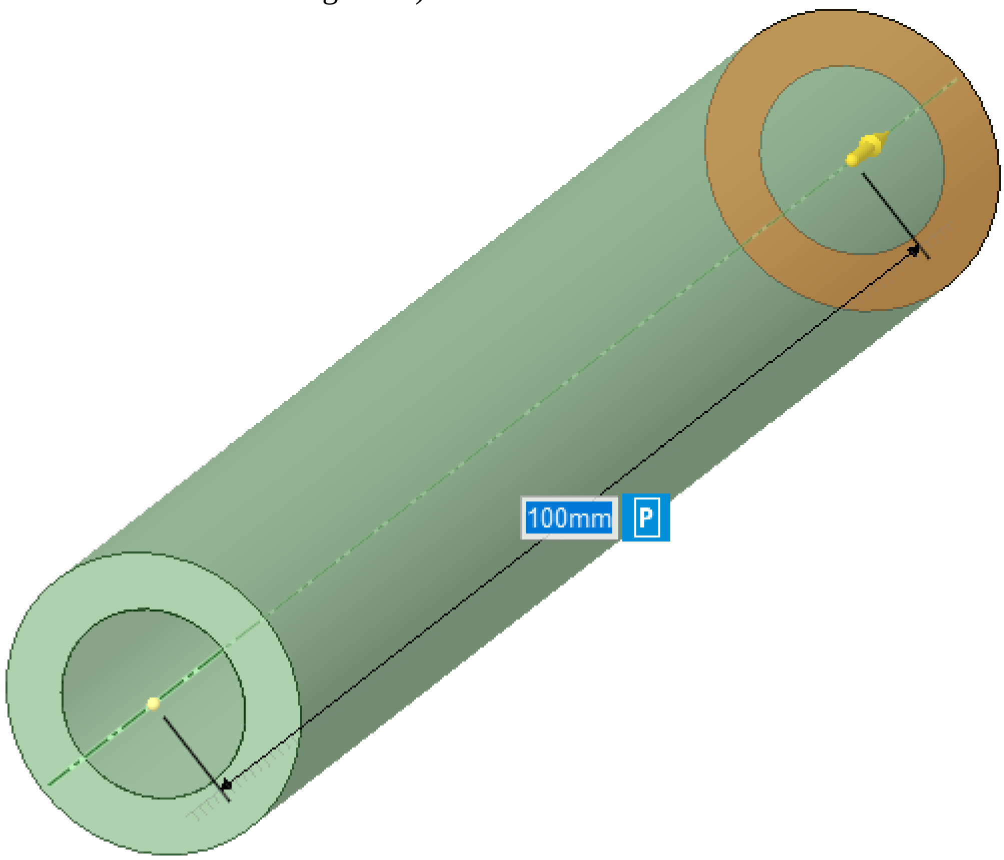




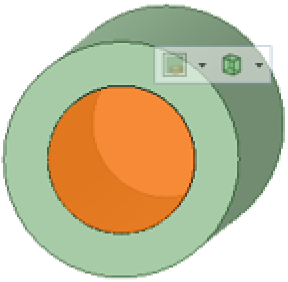
Then position the cursor as shown below



By moving the cursor while pressing LMB you will notice the changing dimension of the pipe length. Enter 100 mm and confirm with Enter (you may have to do it while holding LMB).



The last step is to remove the unnecessary plane sealing the pipe opening. To do this, press *Esc* and then select the unnecessary plane (it will turn orange as in the figure below) and press *Delete*.



4) Close *Spaceclaim* save project in *Workbench* via *Ctrl + s*
5) In the meshing module, make a numerical grid so that it looks similar to the one shown in Fig. 3.

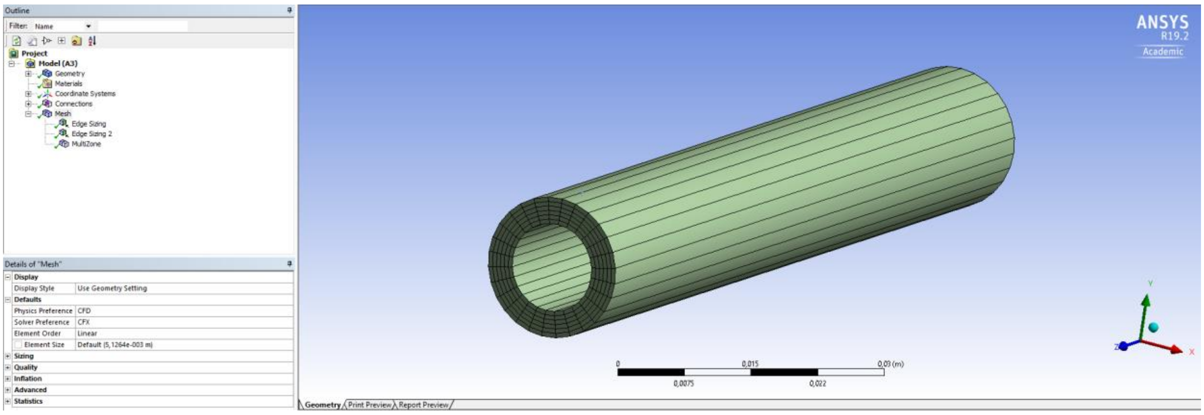
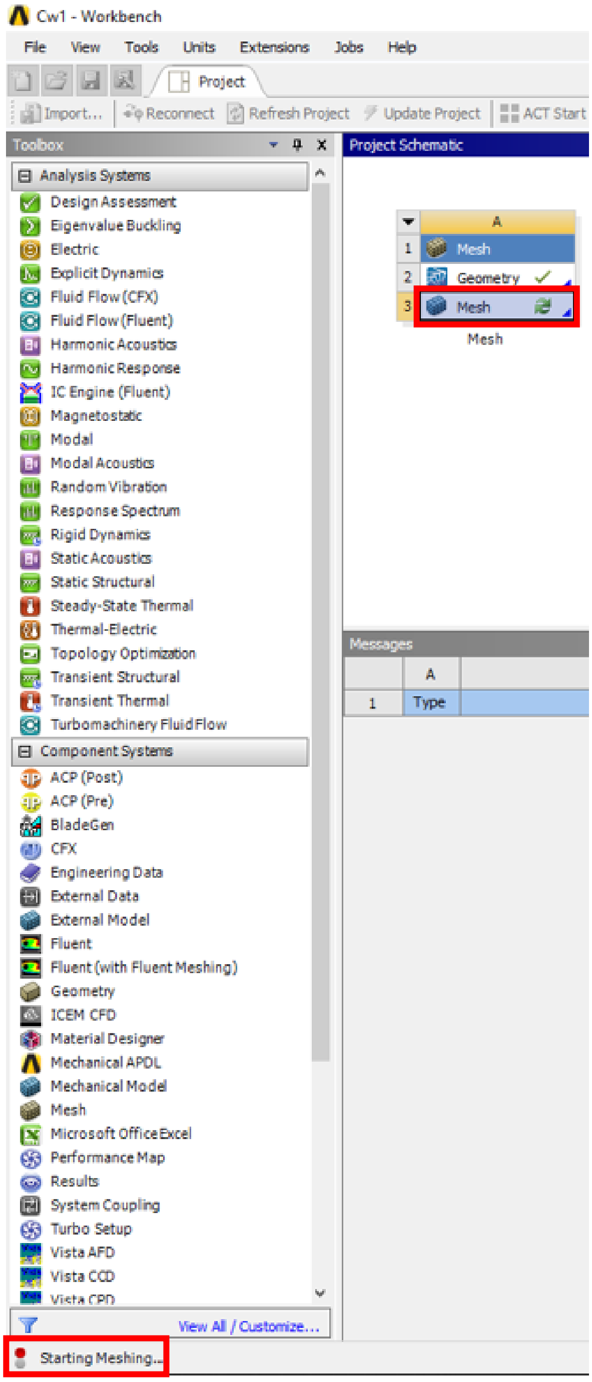
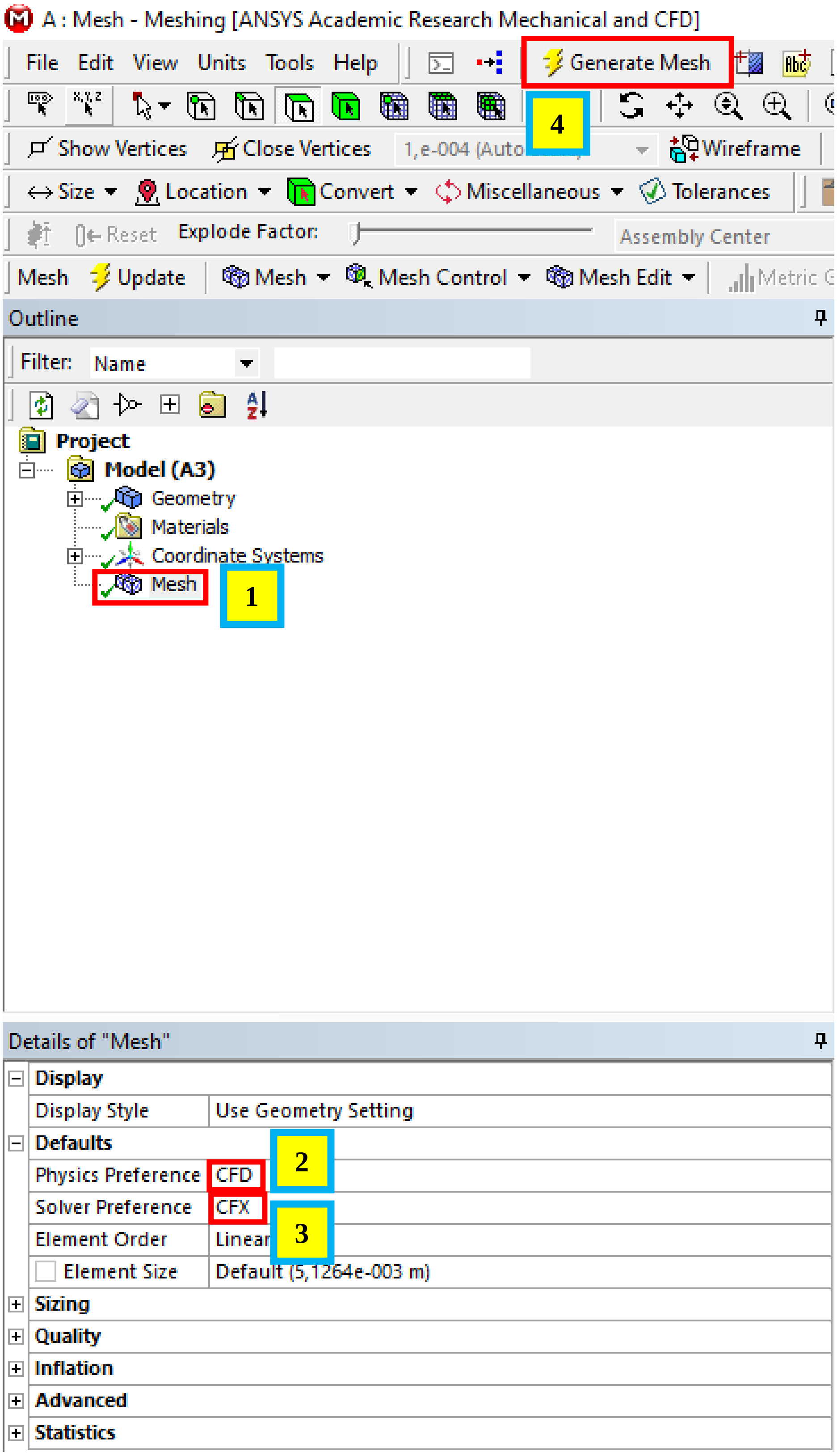


Fig. 3. View of the numerical grid for the problem of one-dimensional transient heat conduction in a cylindrical tube

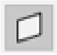
To do this, open Ansys Meshing by double-clicking LMB *Mesh*

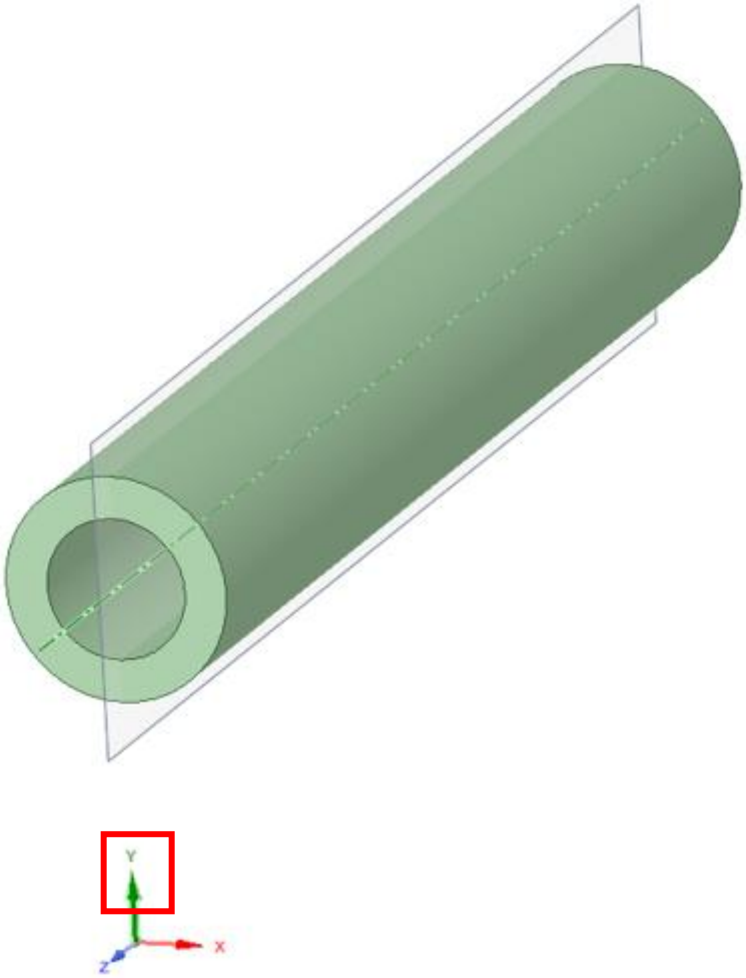


In *Ansys Meshing*: 1) click *Mesh*, 2) change *Physisc Preference* into *CFD*, 3) Change *Solver Preference* into *CFX*, 4) Click LMB *Generate Mesh*

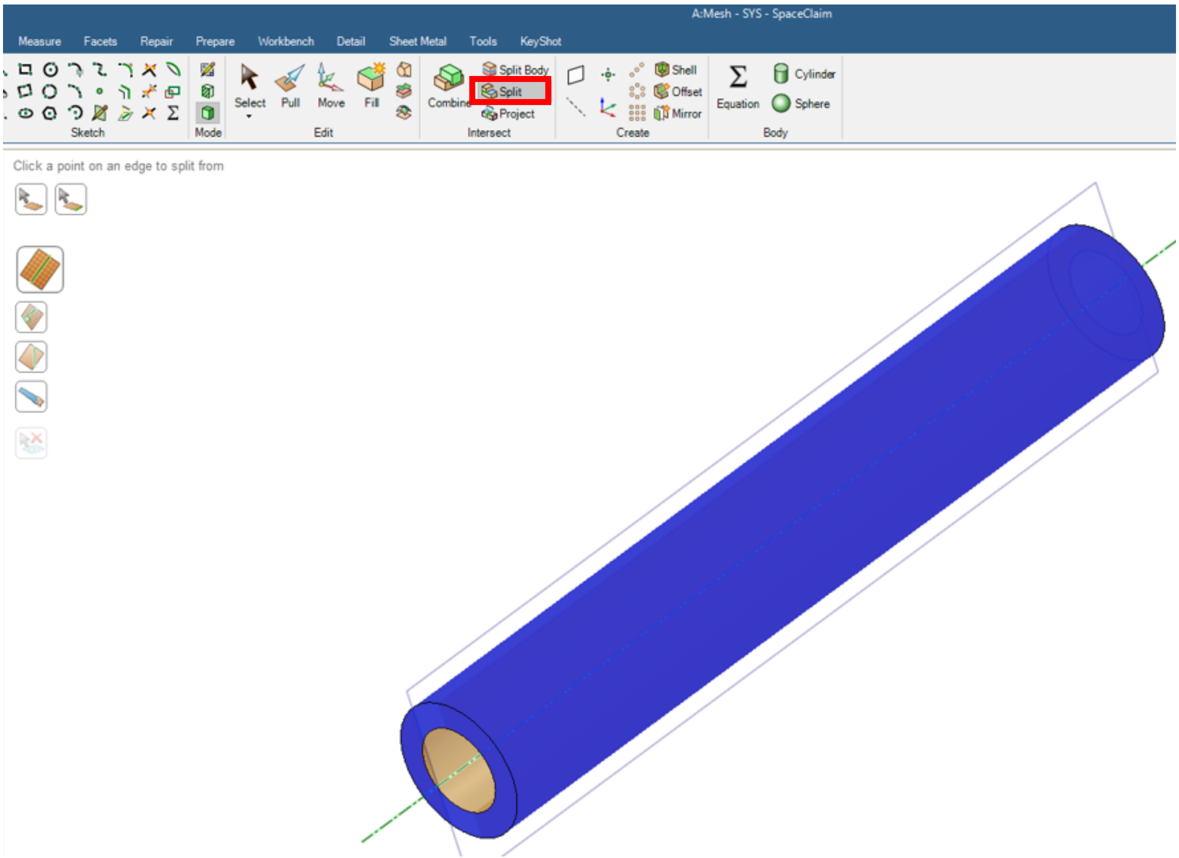


As you can see, the default mesh is not similar to the mesh in Fig. 3. To fix it, you must first edit the geometry. We will cut the pipe in half to create edges, on which then it will be possible to give the number of elements of the numerical grid. Close the *Ansys Meshing* module and open the program *Spaceclaim*.

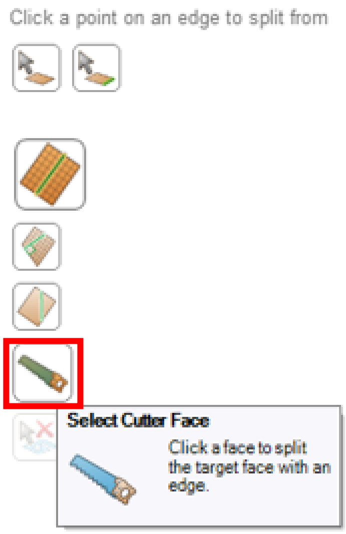
Select the plane creation icon  and LMB click the Y axis to create the YZ plane.



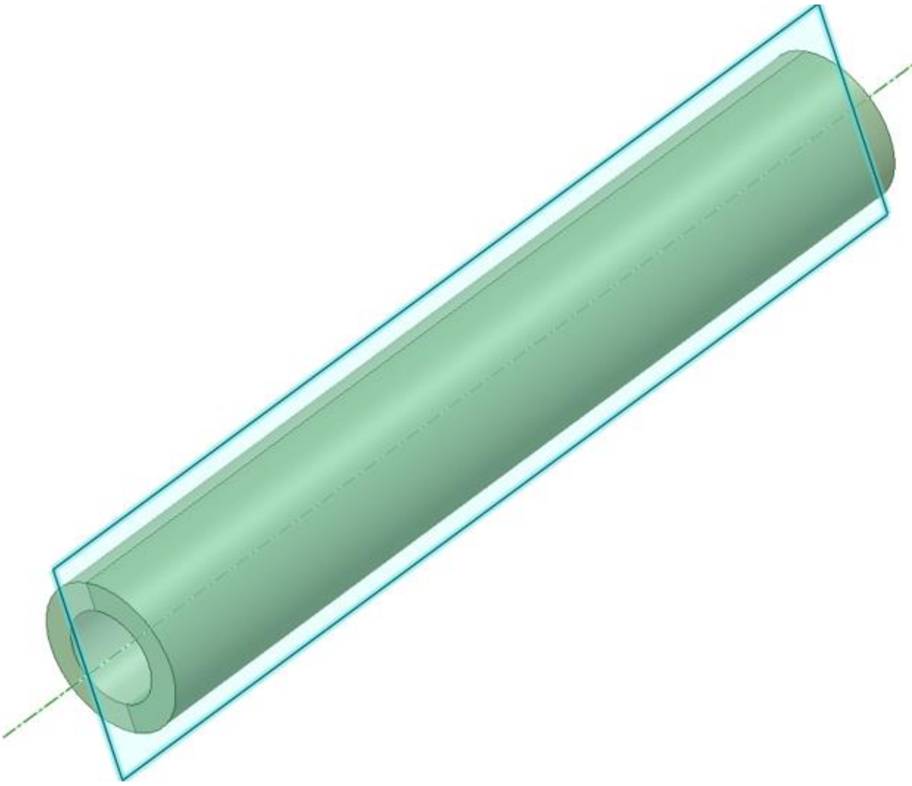
Select *Split* and indicate LMB with the *Ctrl* key pressed down 3 outer surfaces of the pipe (marked in blue)



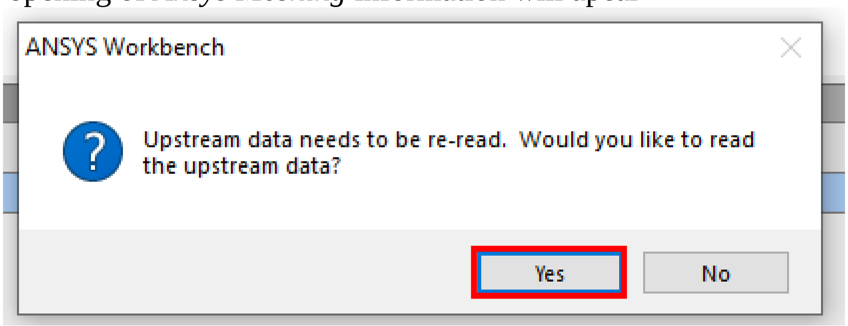
Next choose *Select Cutter Face*



And then point to the YZ plane you created earlier and close the program *Spaceclaim* and open *Ansys Meshing.*

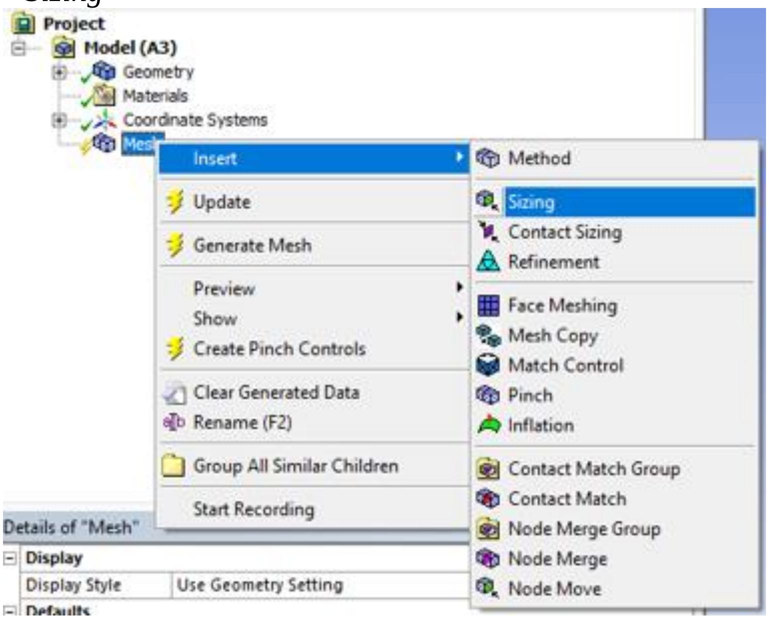


During opening of *Ansys Meshing* information will apear



Select *Yes*.

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



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

Select the long edge (dashed line in the figure below) and confirm *Geometry->Apply* (if you do not 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	

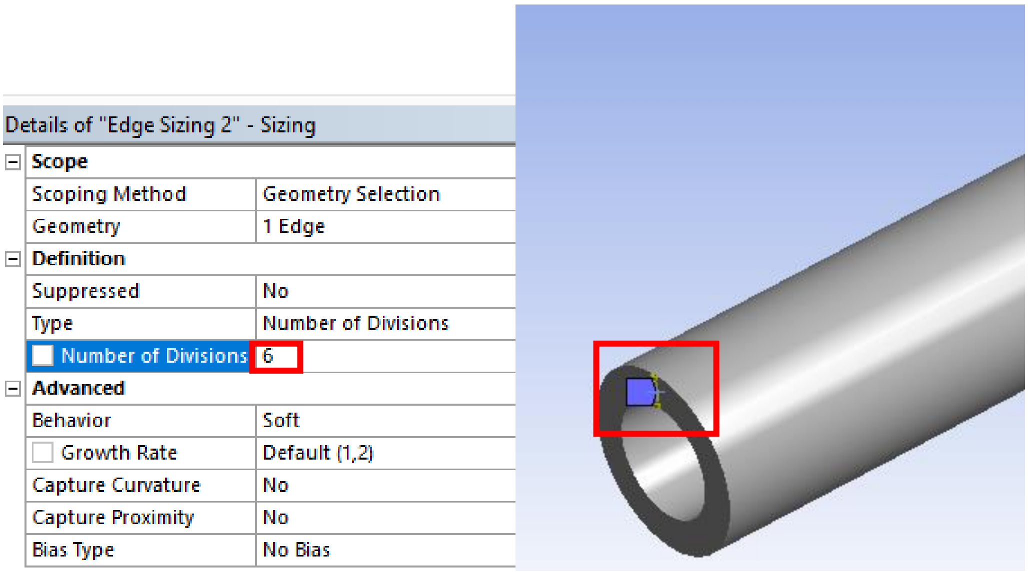
Change *Definition->Type* from *Element Size* into *Number of Divisions*. In the field *Number of Divisions* set value of 1

Details of "Edge Sizing" - Sizing

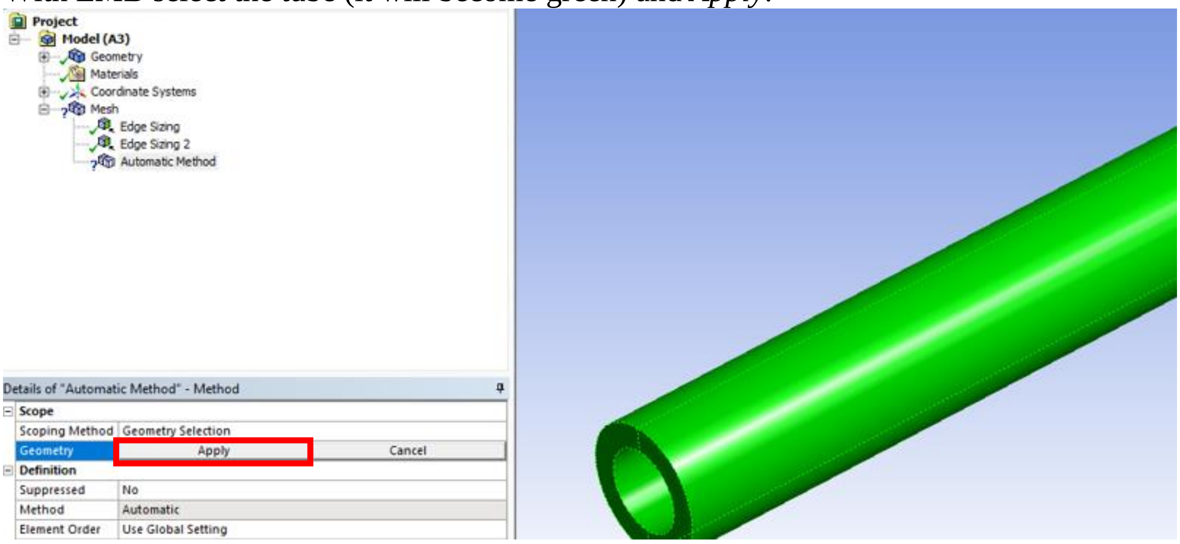
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	1
Advanced	
Behavior	Soft
Growth Rate	Default (1,2)
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

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

Perform similar operations for the short edge shown in the figure below, except that *Number of Divisions* = 6



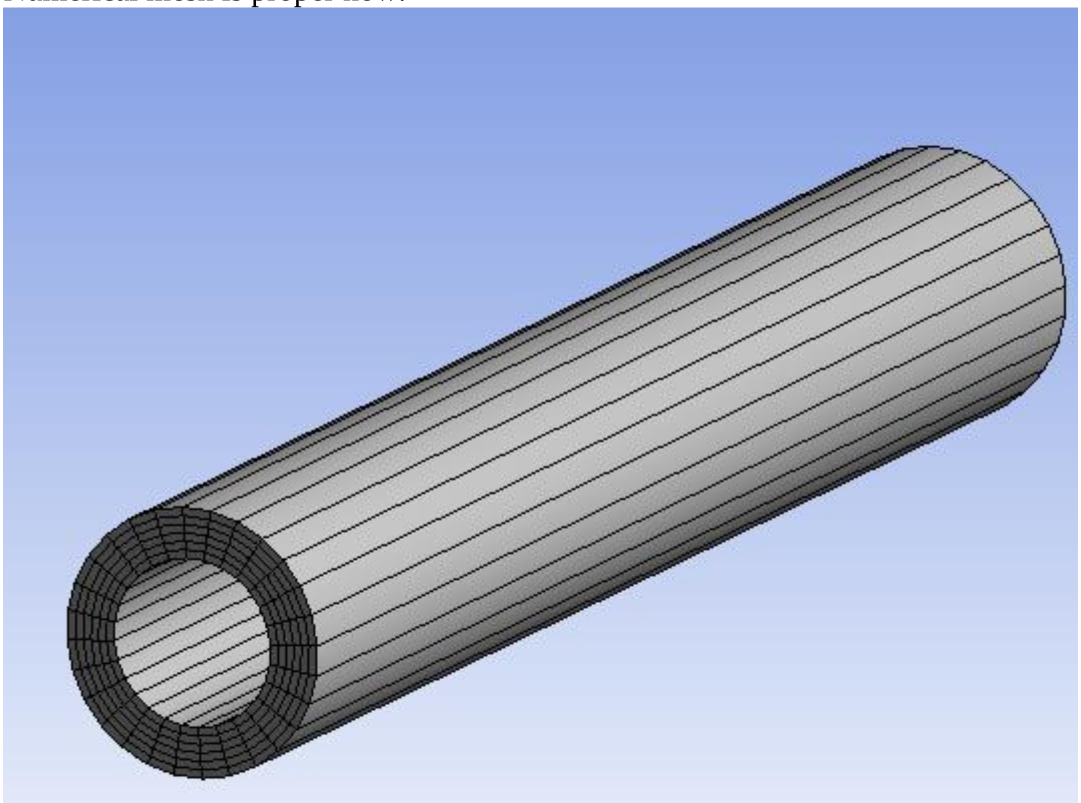
In *Ansys Meshing* press RMB on *Mesh* and select *Insert->Method*
With LMB select the tube (it will become green) and *Apply*.



Change *Method* from *Automatic* into *Multizone* and click *Generate Mesh*

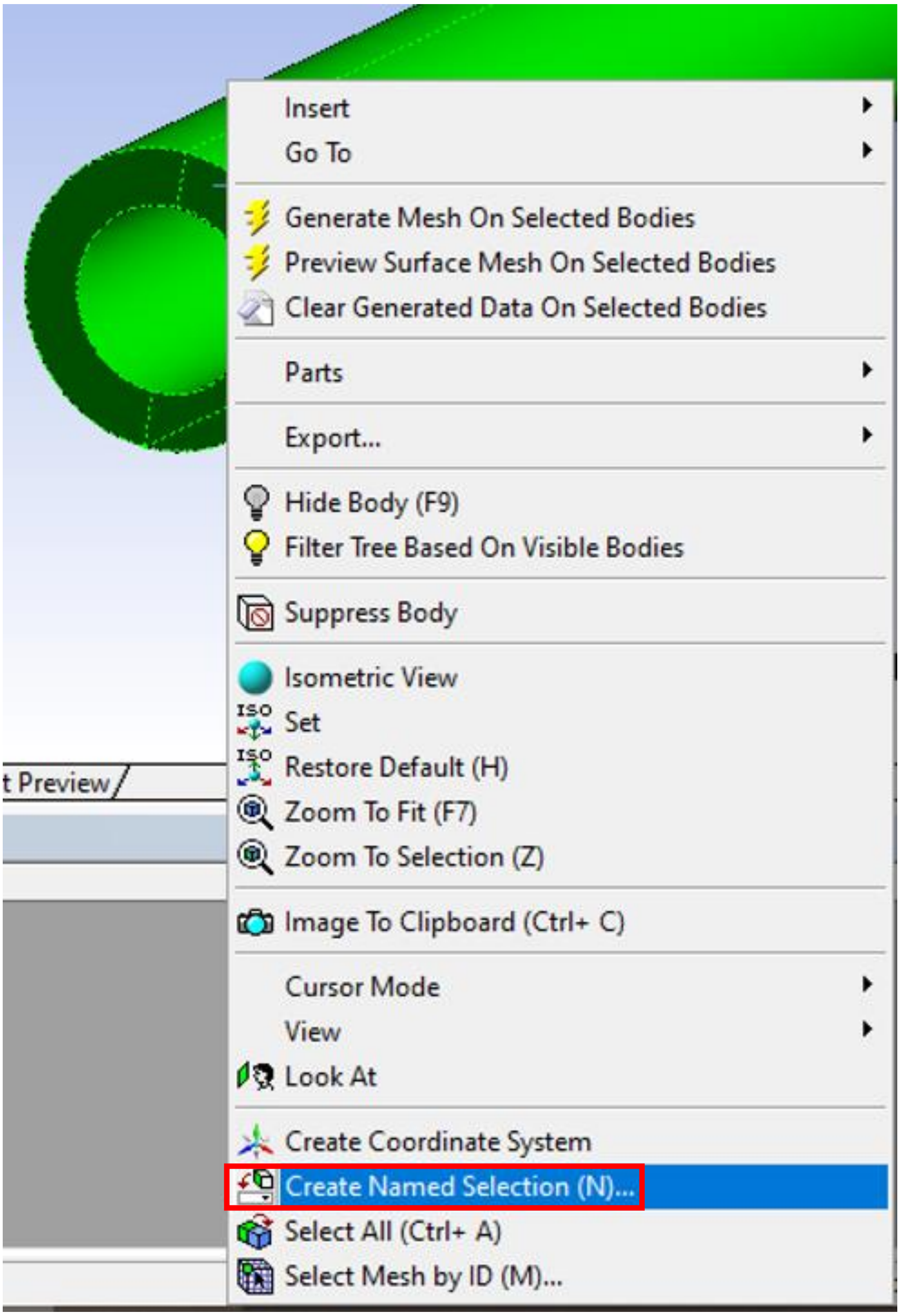
Details of "MultiZone" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Order	Use Global Setting
Src/Trg Selection	Automatic
Source Scoping Method	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	3,e-003 m
Write ICEM CFD Files	No

Numerical mesh is proper now.

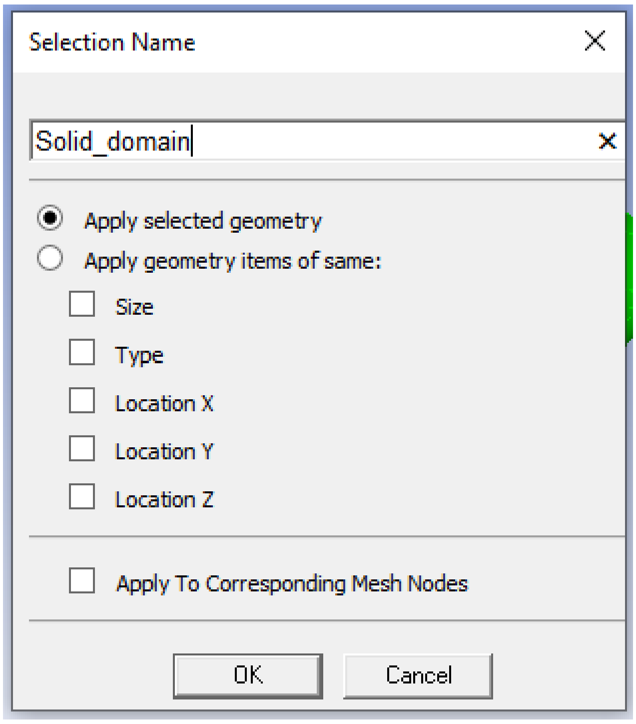


The last step is to name the volumes and surfaces.
Press LMB filter of selecting bodies

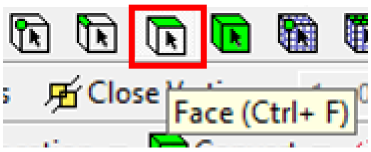
With LMB select the tube and click RMB and select *Create Named Selection*



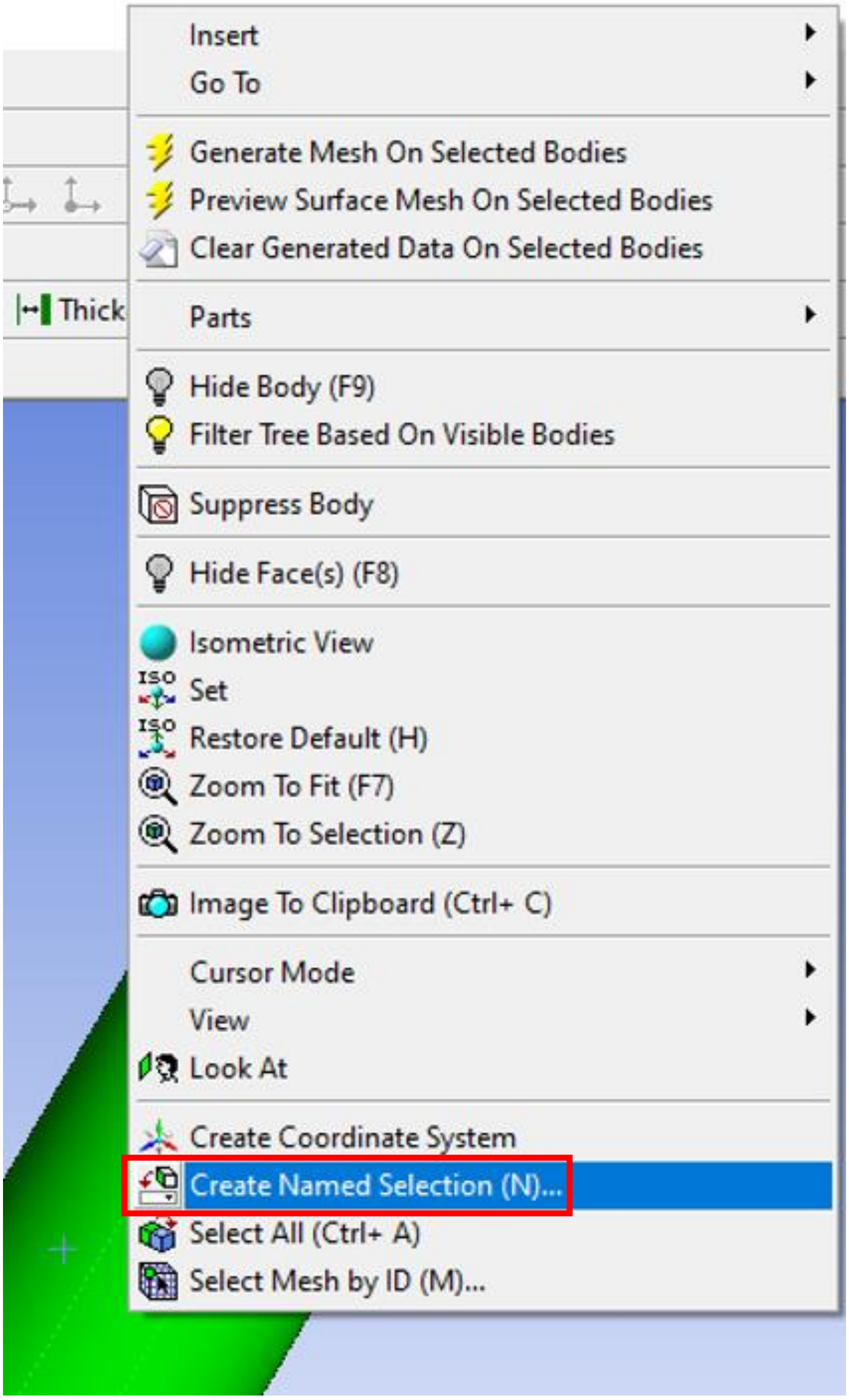
As the name set *Solid_domain*



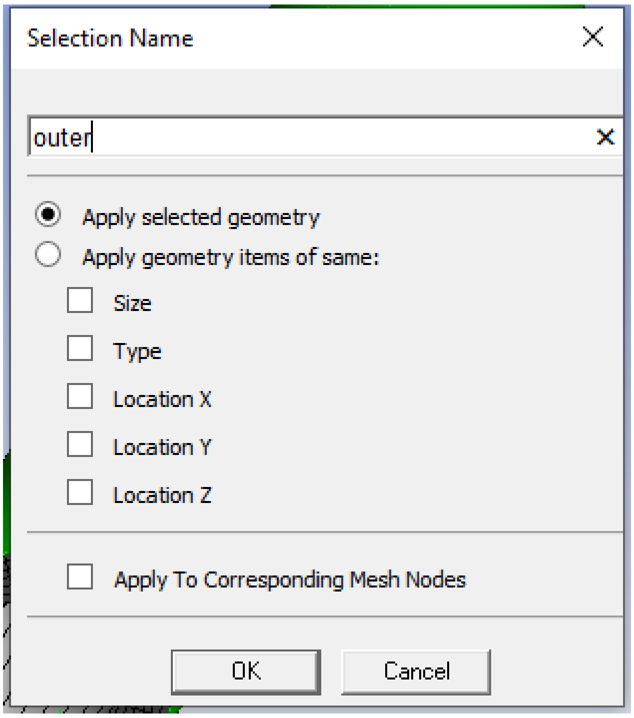
Then change the filter of surface selection



With LMB indicate the outer surface of the pipe and click RMB, then select *Create Named Selection* (the pipe was previously divided in half so now the outer surface also consists of two parts; when choosing the outer surface, select both shelves while holding down the *Ctrl* key).



Set name *outer*



Similarly, create *inner* and *side* names for the inner surface of the pipe and two side surfaces, respectively (for *side* surfaces, select 4 parts).
Close *Ansys Meshing* save project in *Workbench.*

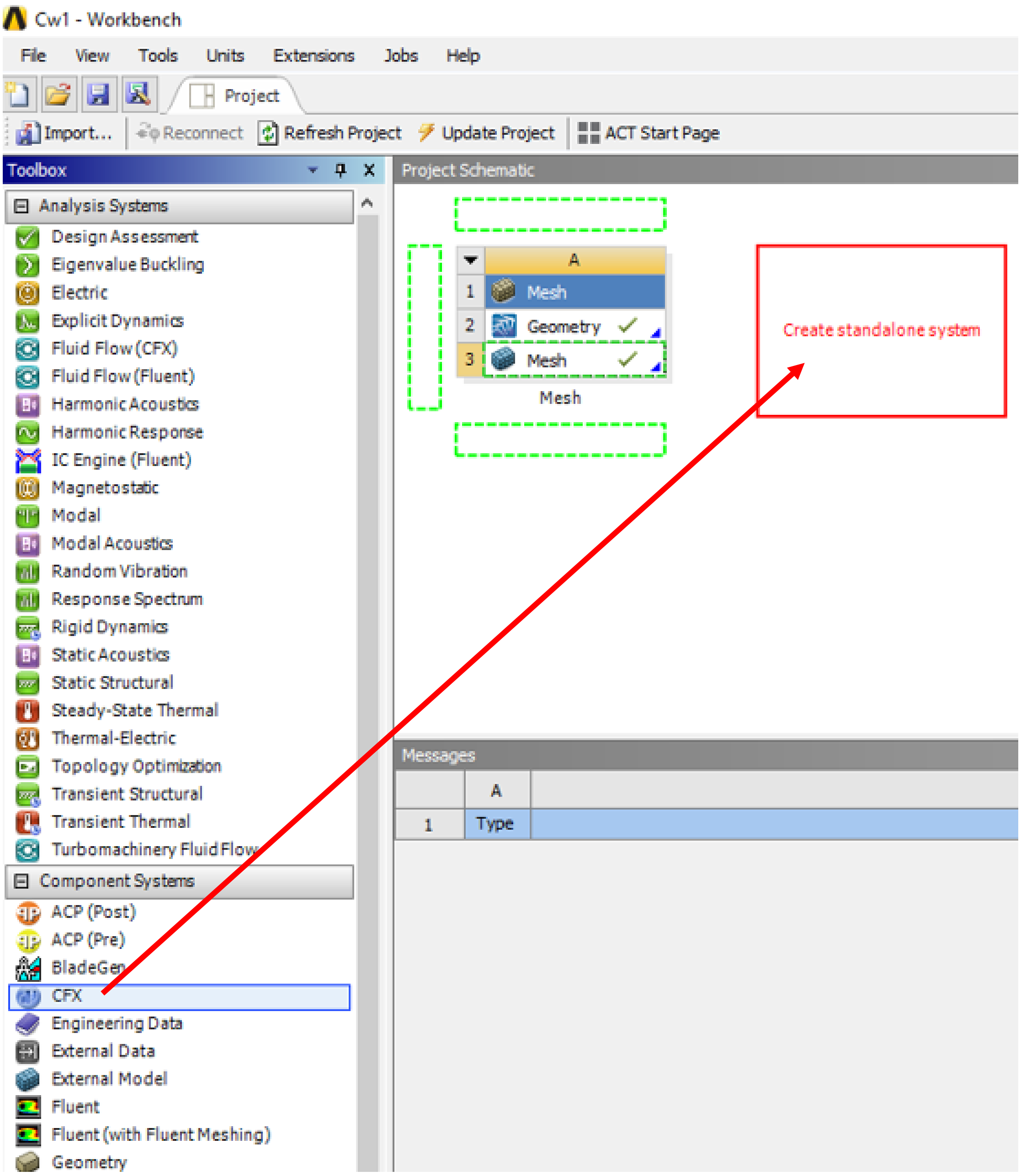
6) In the CFX module, perform calculations of one-dimensional transient heat conduction through a cylindrical wall. Set a constant temperature on the inside of the pipe T_1 = 100°C, while at the outer surface T_2 = 20°C. The thermal conductivity coefficient should be implemented as temperature dependent according to the following formula

$$\lambda = \lambda_0 \left(1 + bT\right) \tag{1}$$

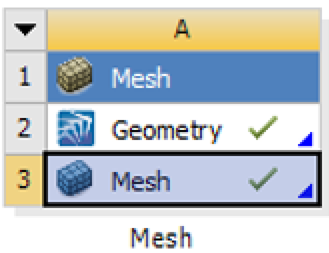
Tab. 1. Parameters used in calculations

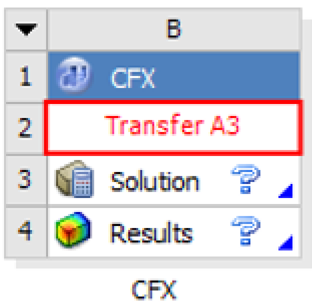
λ_0 W/(m K)	b 1/K	t s	dt s
40	-0,001	2	0.05

To do this, in *Project Schematic* drag the LMB on the *CFX* module

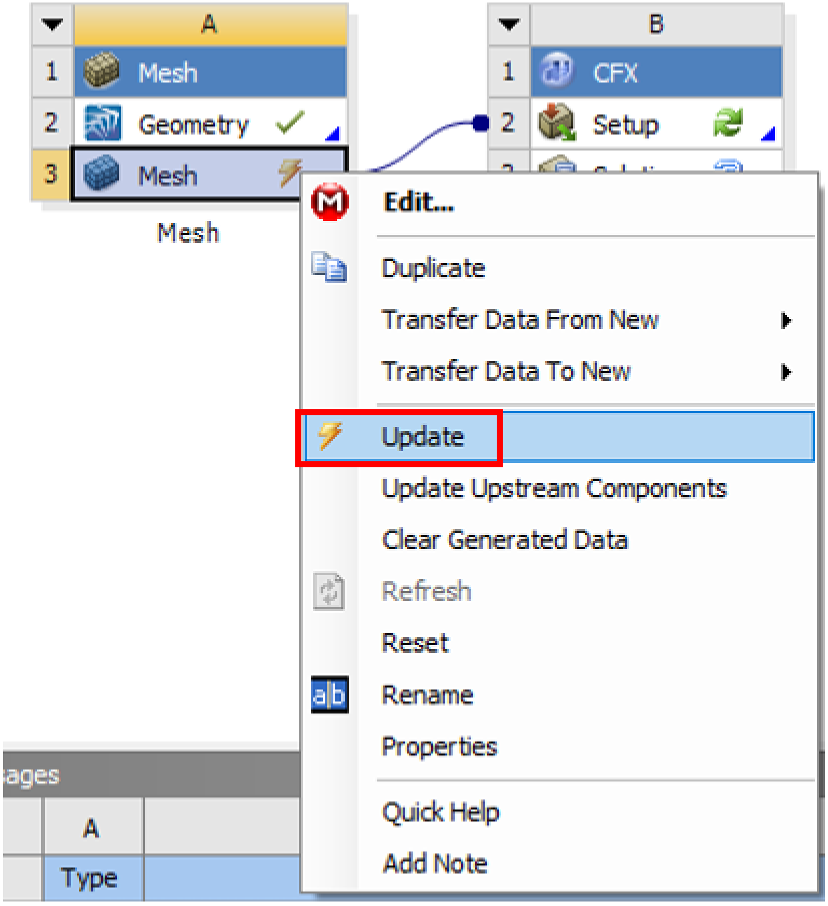


To connect the *Mesh* module with *CFX*, grab LMB *Mesh* (the second one) and drag it to *Setup* until the *Transfer A3* box appears, then release LMB - the connection has been created

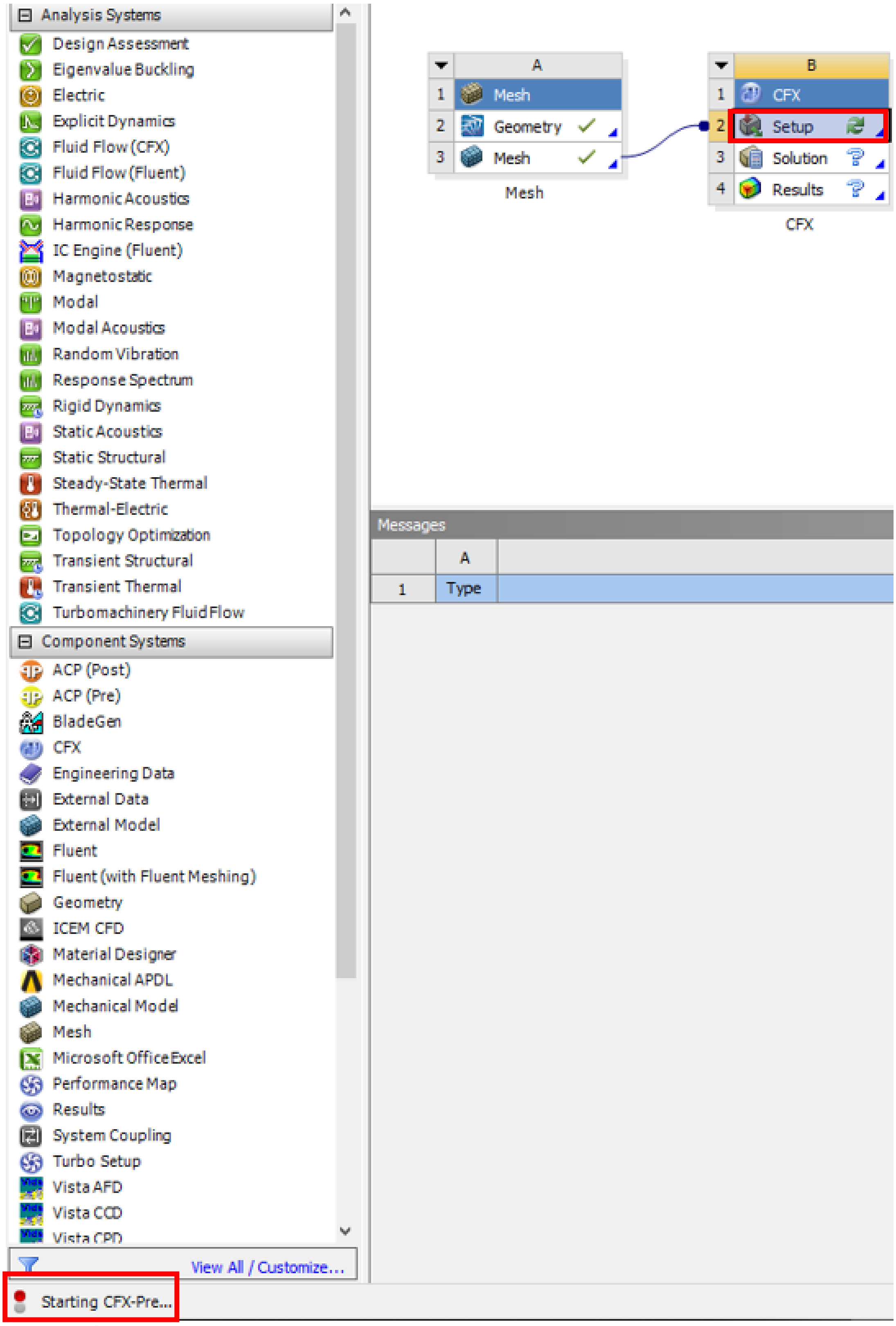




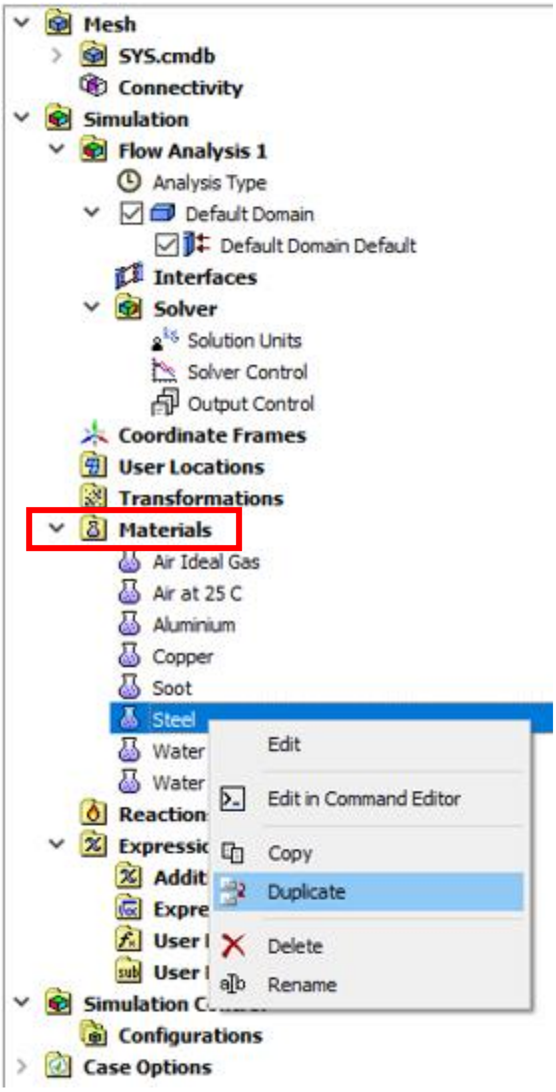
Click RMB on *Mesh* and select *Update*



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

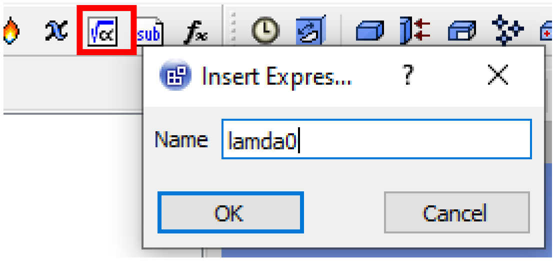


In the first step, you will create a steel material with a heat conduction coefficient that is linearly dependent on temperature according to equation (1). To do this, expand *Materials* in the tree and press RMB on *Steel* material, then select *Duplicate* to create a copy.

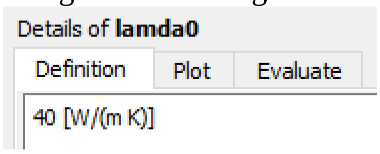


Change name from *Copy of Steel* into *SteelLinearThermCond*

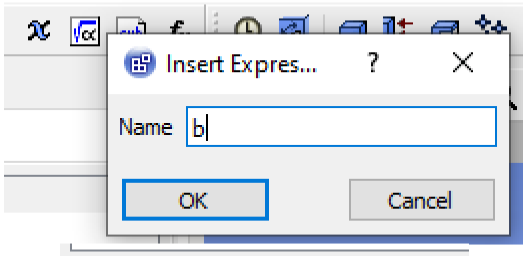
Create *Expression* with the name *lambda0*

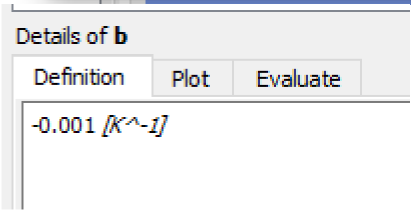


In *Details* field enter according to the drawing below and confirm *Apply*.

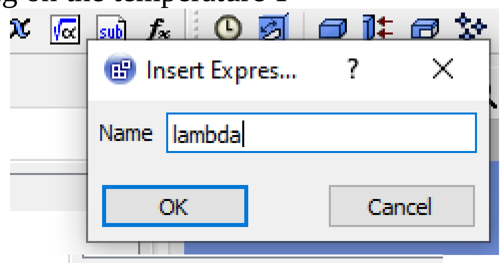


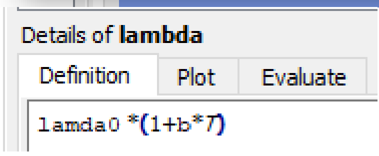
Create another expression with the name *b*



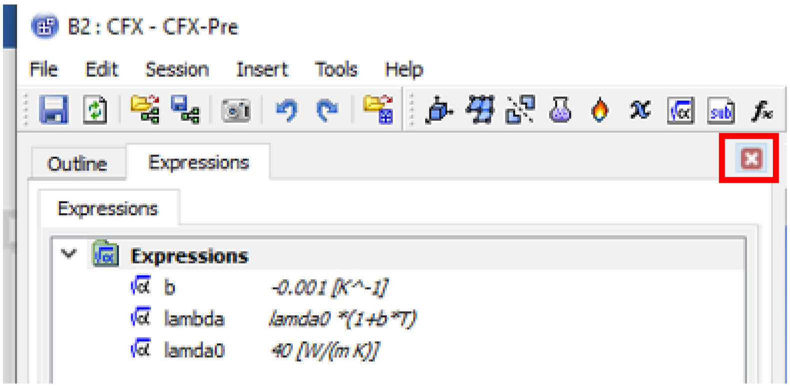


Create an expression called *lambda* that calculates the thermal conductivity coefficient depending on the temperature *T*

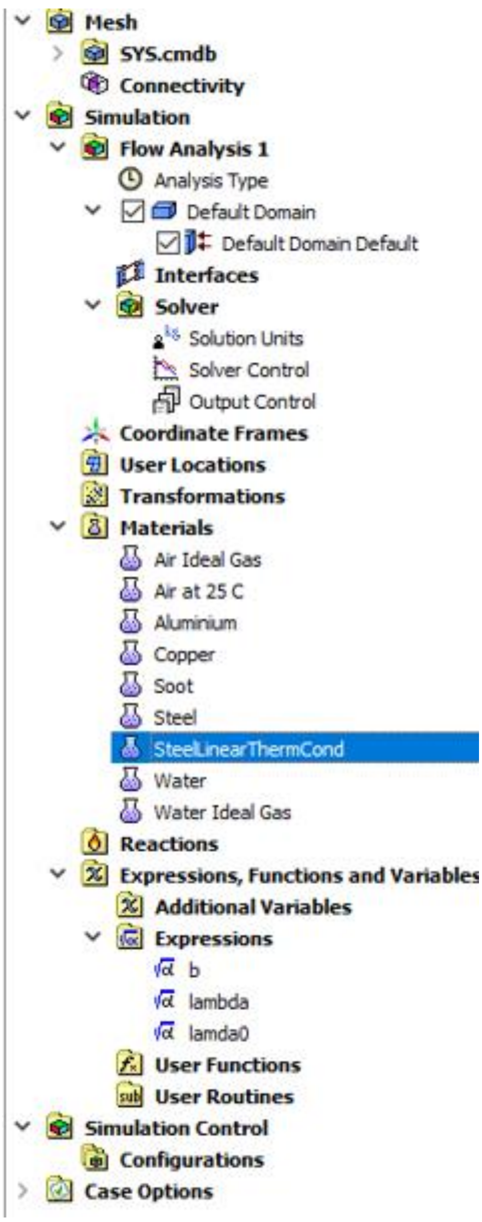




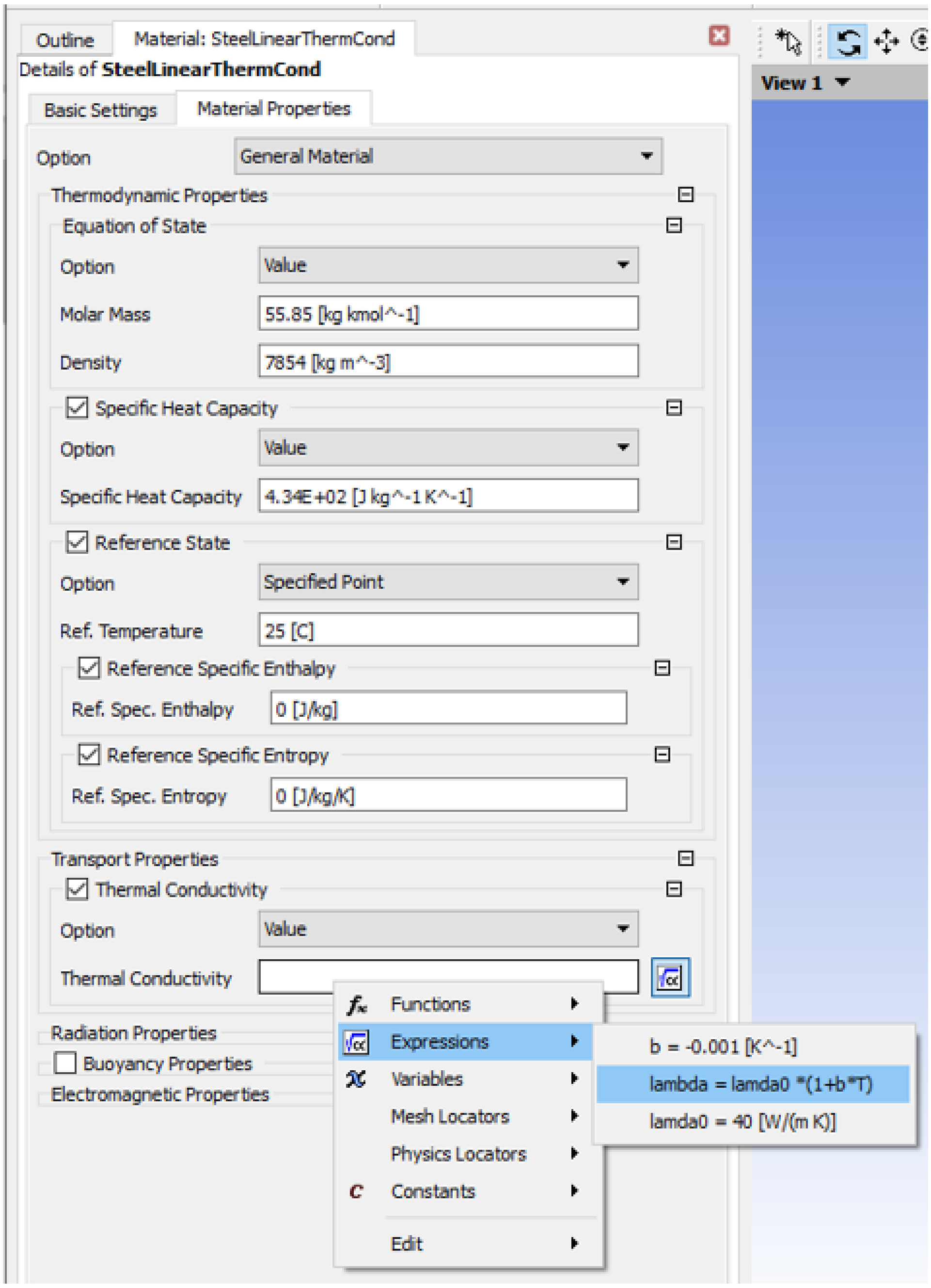
Close expressions editor



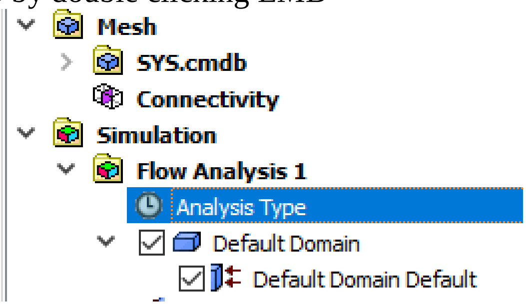
Double click LMB on *SteelLinearThermCond* to edit it



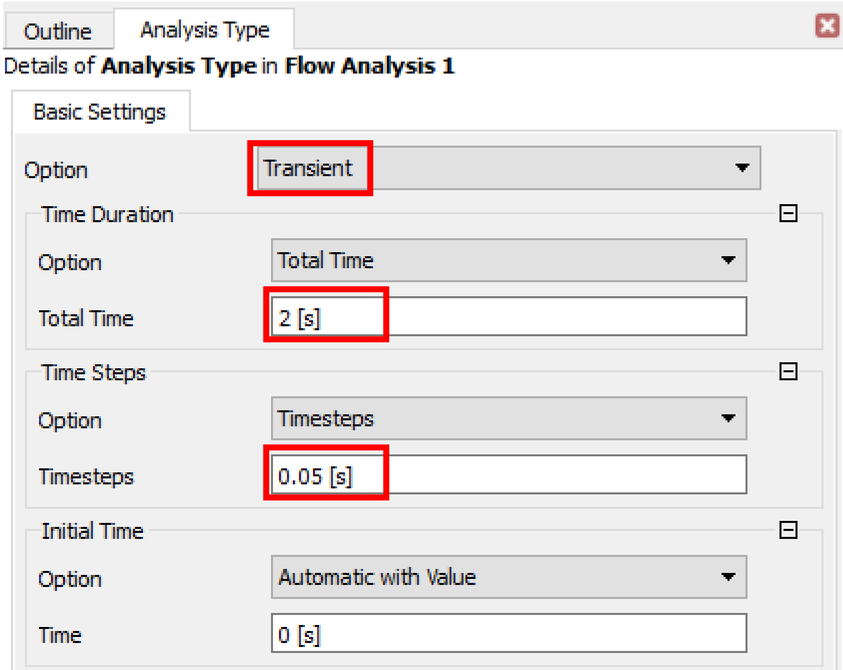
In tab *Material Properties* expand *Transport Properties* and in the field *Thermal Conductivity* click *Expressions*  icon and next click RMB in the field *Thermal Conductivity* and select *Expressions->lambda*. Confirm *OK*.



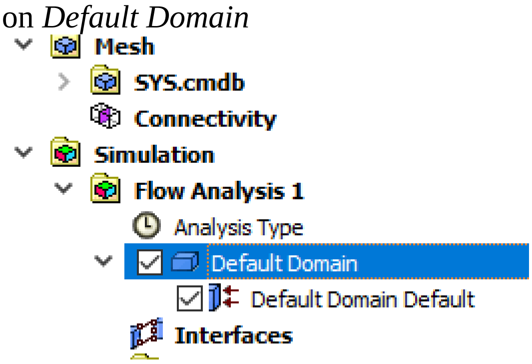
Open *Analysis Type* by double clicking LMB



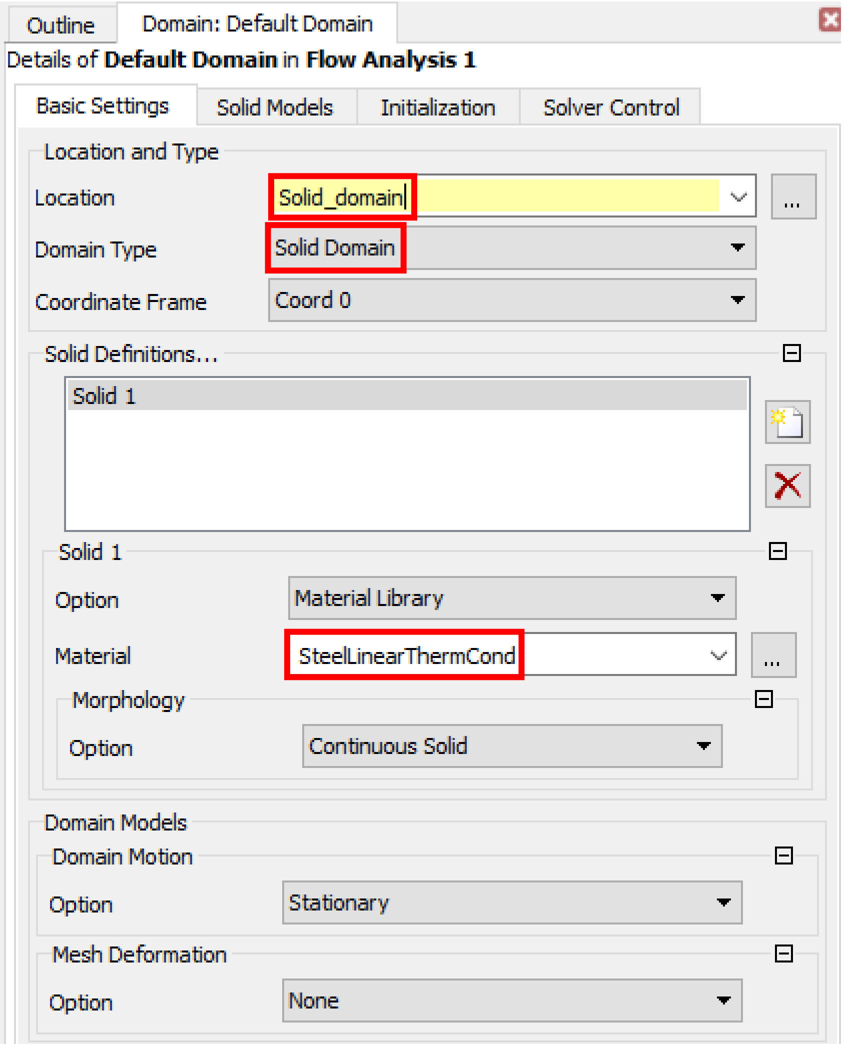
Apply the following settings according to tab. 1 and confirm *OK*.



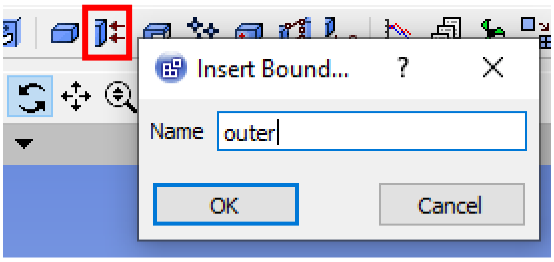
Double click LMB on *Default Domain*



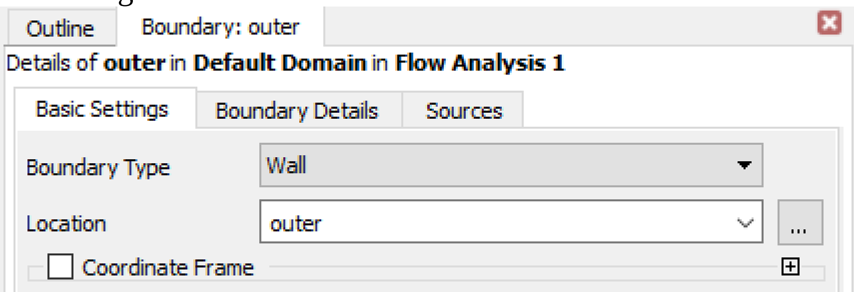
Apply the following settings and confirm *OK*.

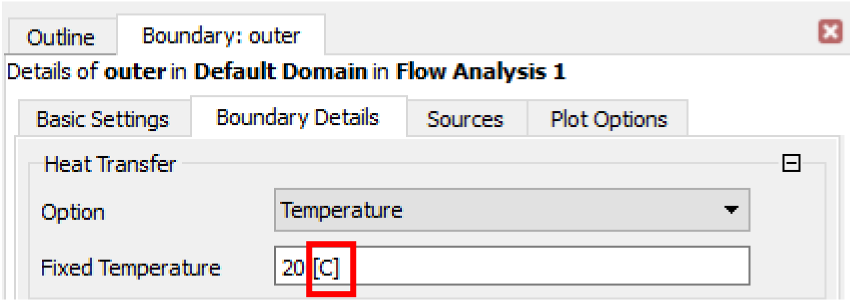


Create a boundary condition named *outer*

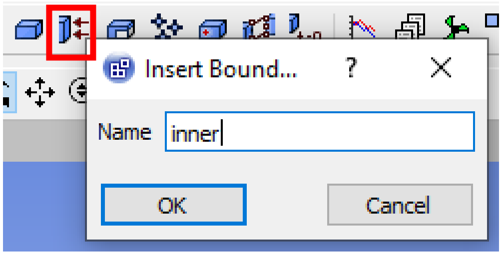


Apply the settings as below

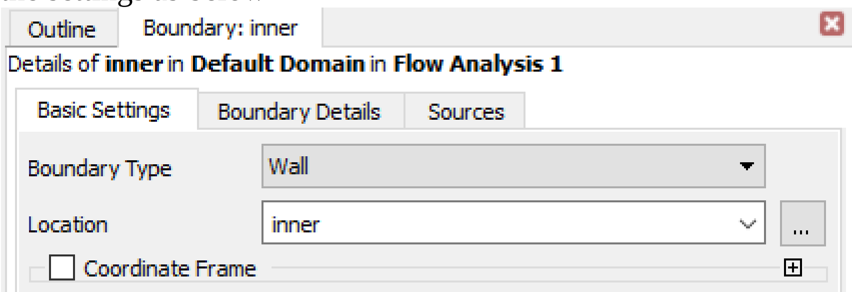


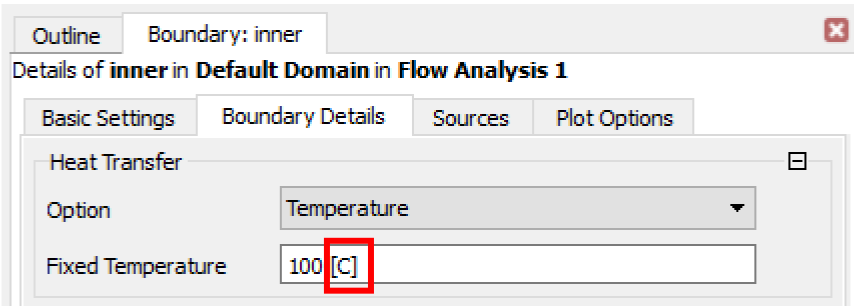


Create a boundary condition named *inner*

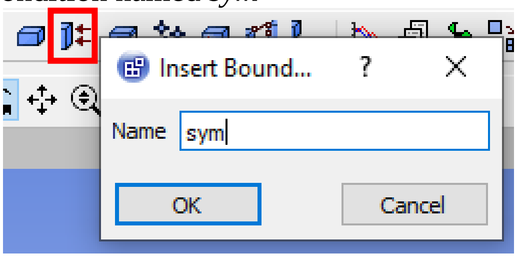


Apply the settings as below

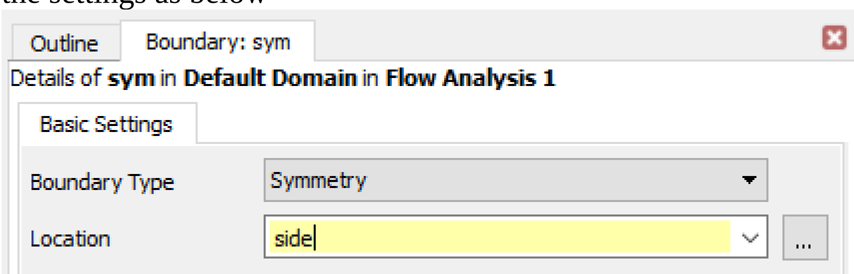




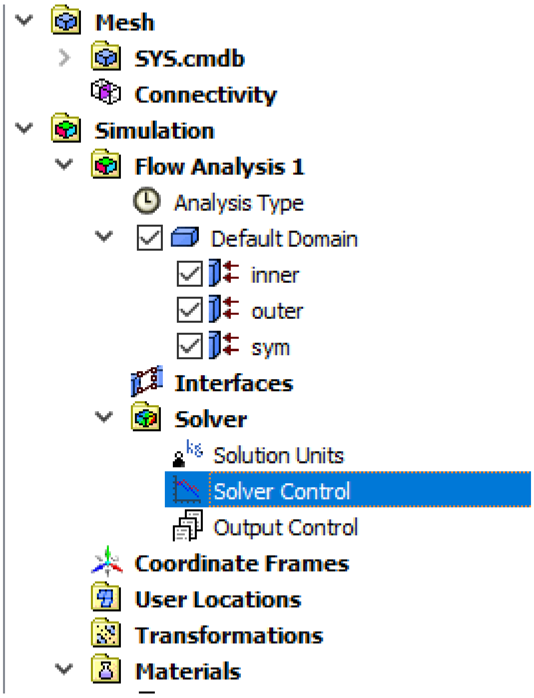
Create a boundary condition named *sym*



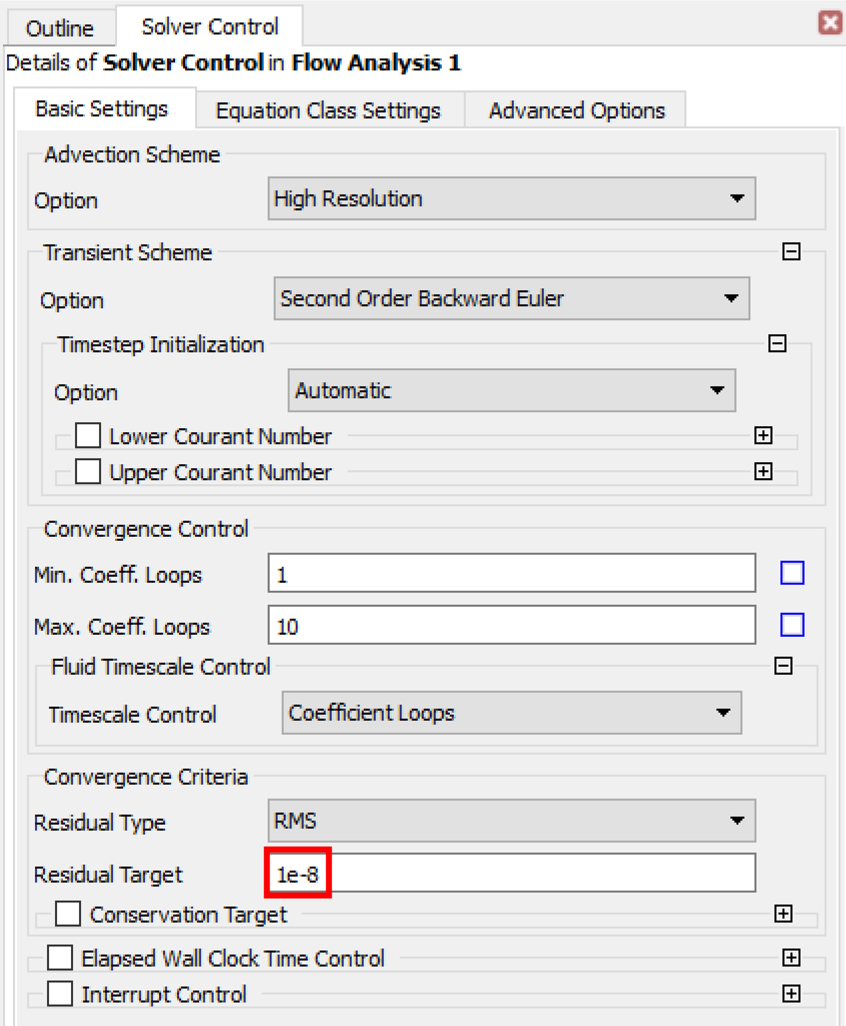
Apply the settings as below



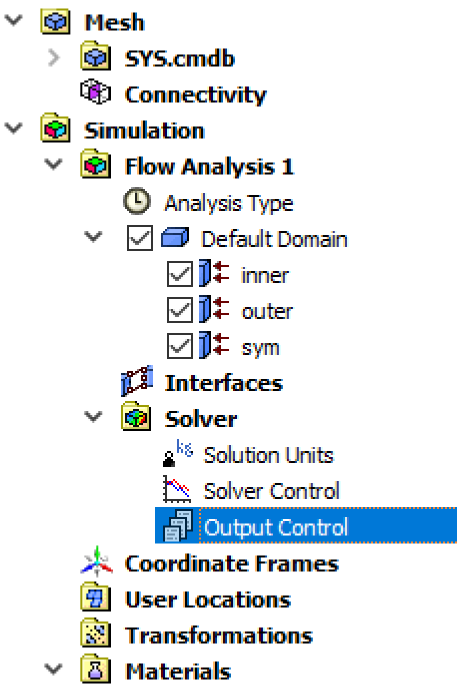
Open *Solver Control*



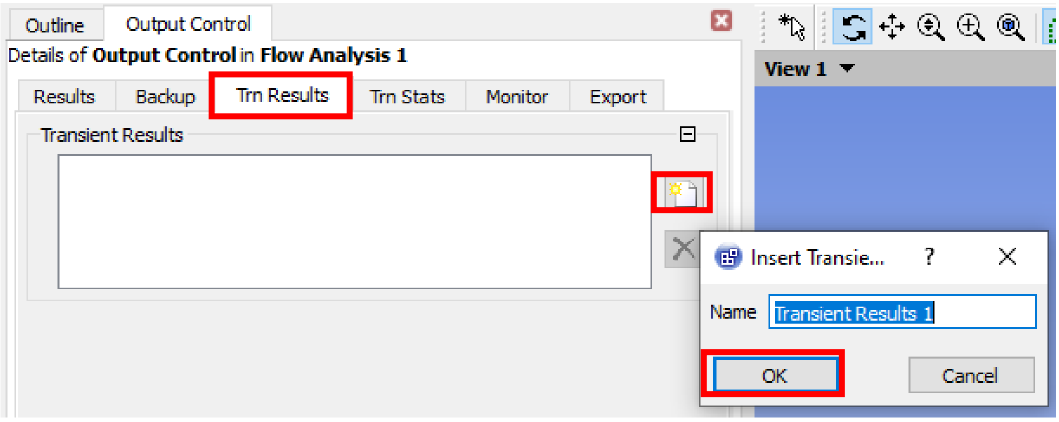
Apply the following settings and confirm *OK*.

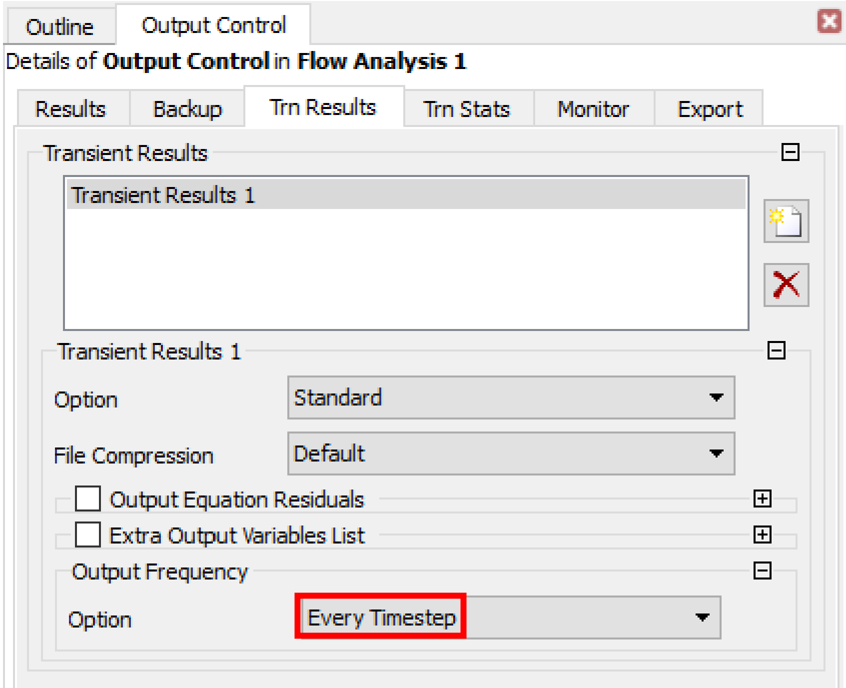


Open *Output Control*



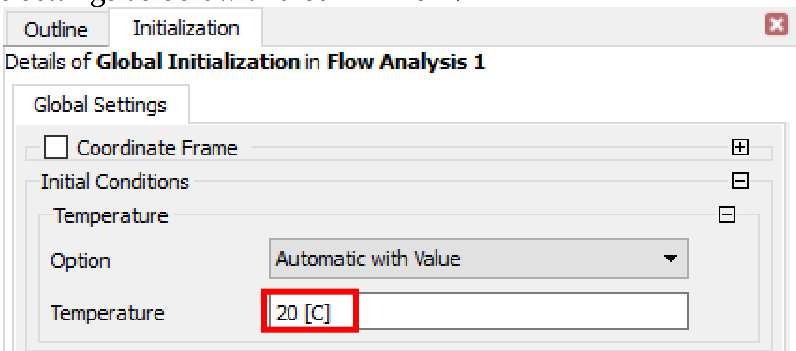
Apply the following settings and confirm *OK*.



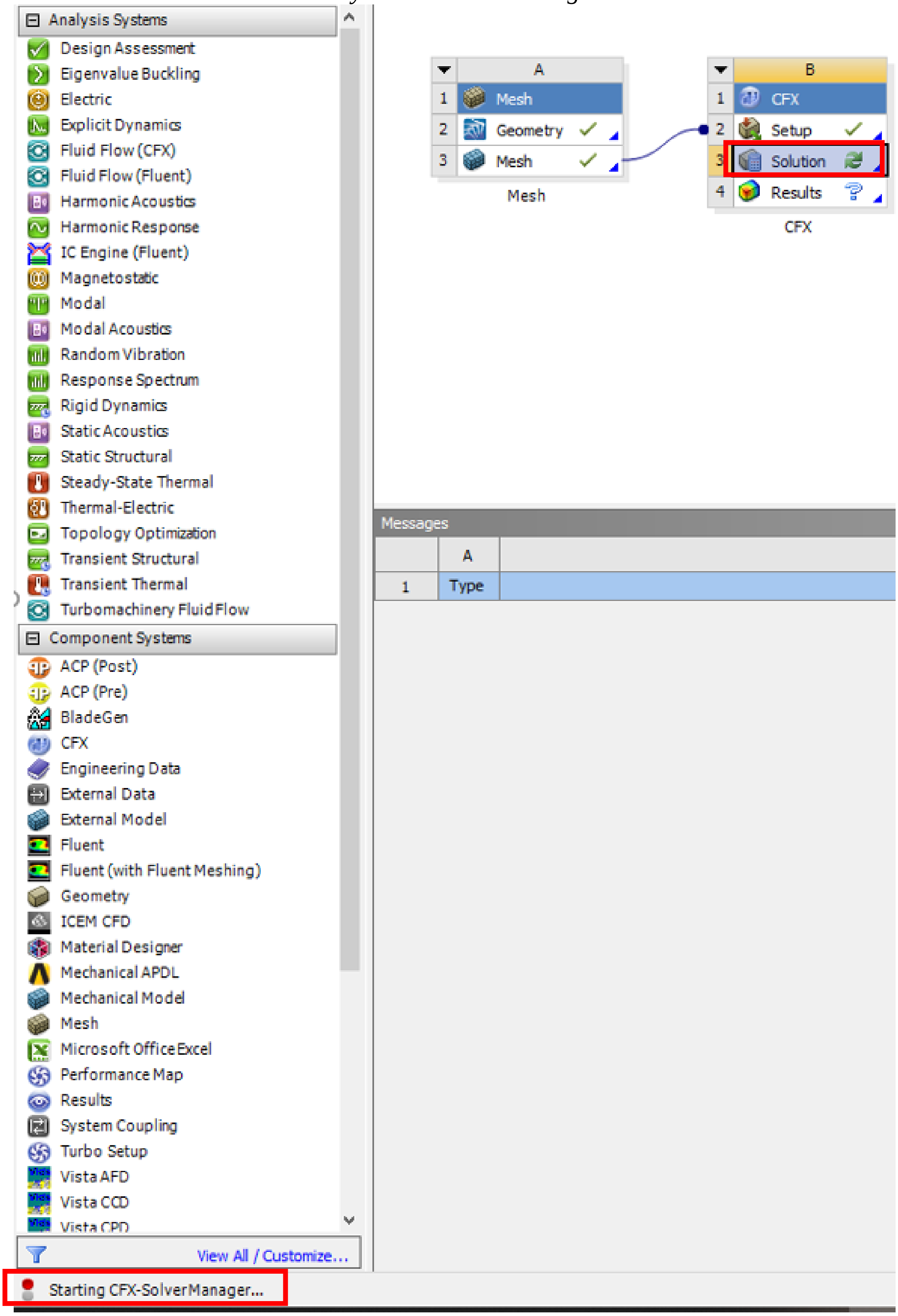


Create initial conditions by clicking the LMB on *Global Initialization* icon (at the top of the screen, almost in the middle)

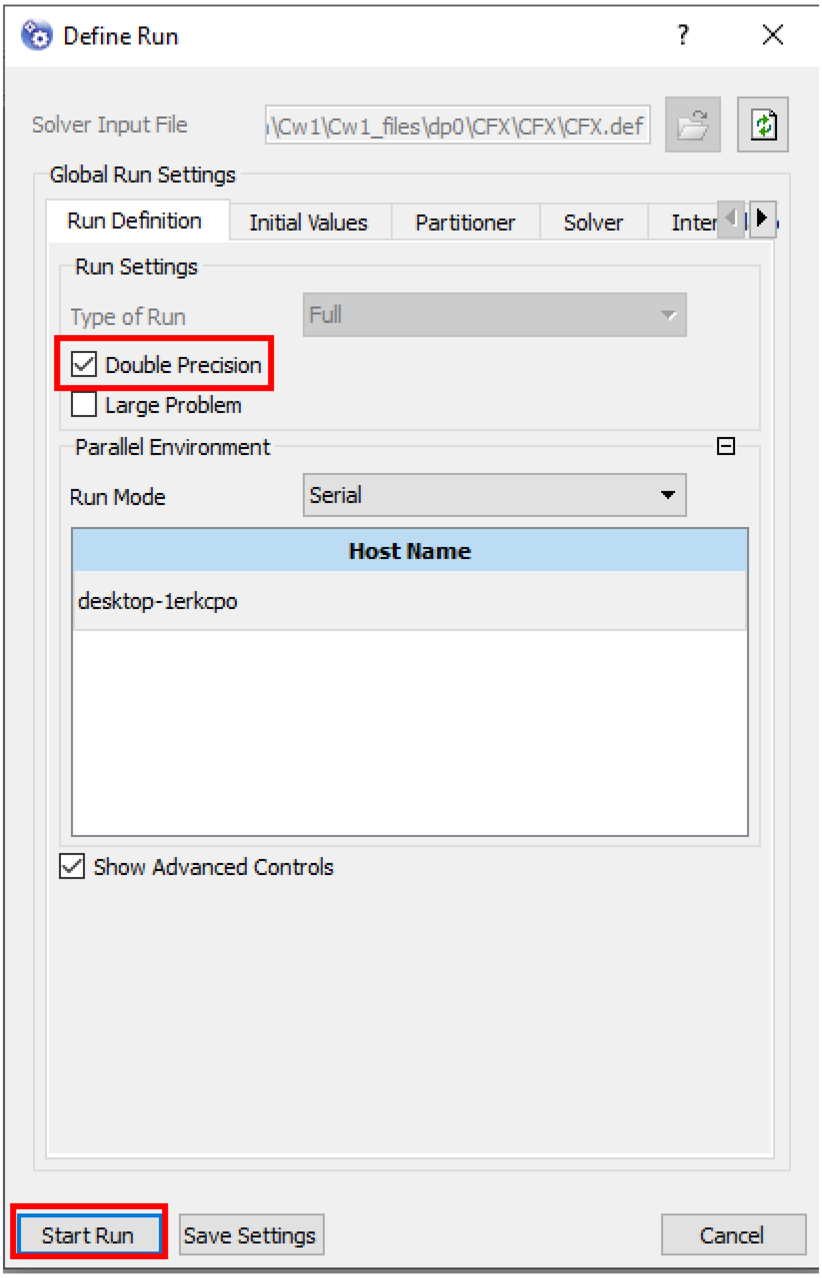
Apply the settings as below and confirm *OK*.



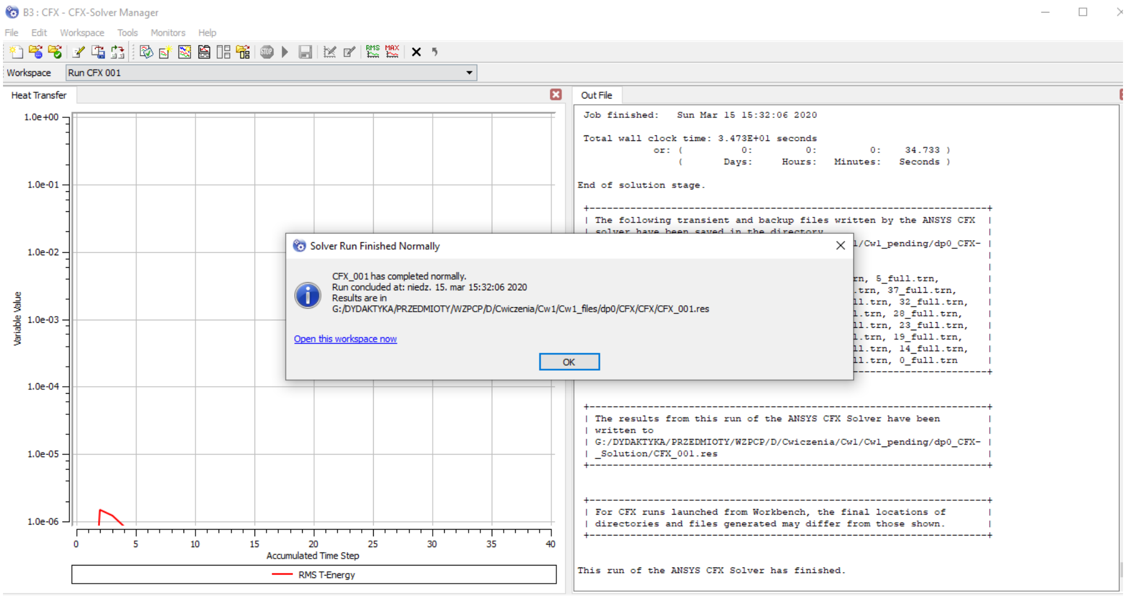
Close *Ansys CFX*.
Double click *Solution* to run *Ansys CFX Solver Manager*



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



After completing the calculations, the program will display a message:



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

7) The report should present the contours of the temperature distribution in the cylindrical partition for times: 0; 0,1; 0,2; 0,3; 0,4; 0,5; 1,0; 2,0 s. In addition, the numerical solution of the wall temperature distribution for 2 s should be compared with the analytical solution on one chart (2)

$$T(r)=\frac{1}{b}\left[\sqrt{\left(\frac{\lambda_1}{\lambda_0}\right)^2-\frac{\dot{q}_L b}{\pi\lambda_0}\ln\frac{r}{r_1}}-1\right] \tag{2}$$

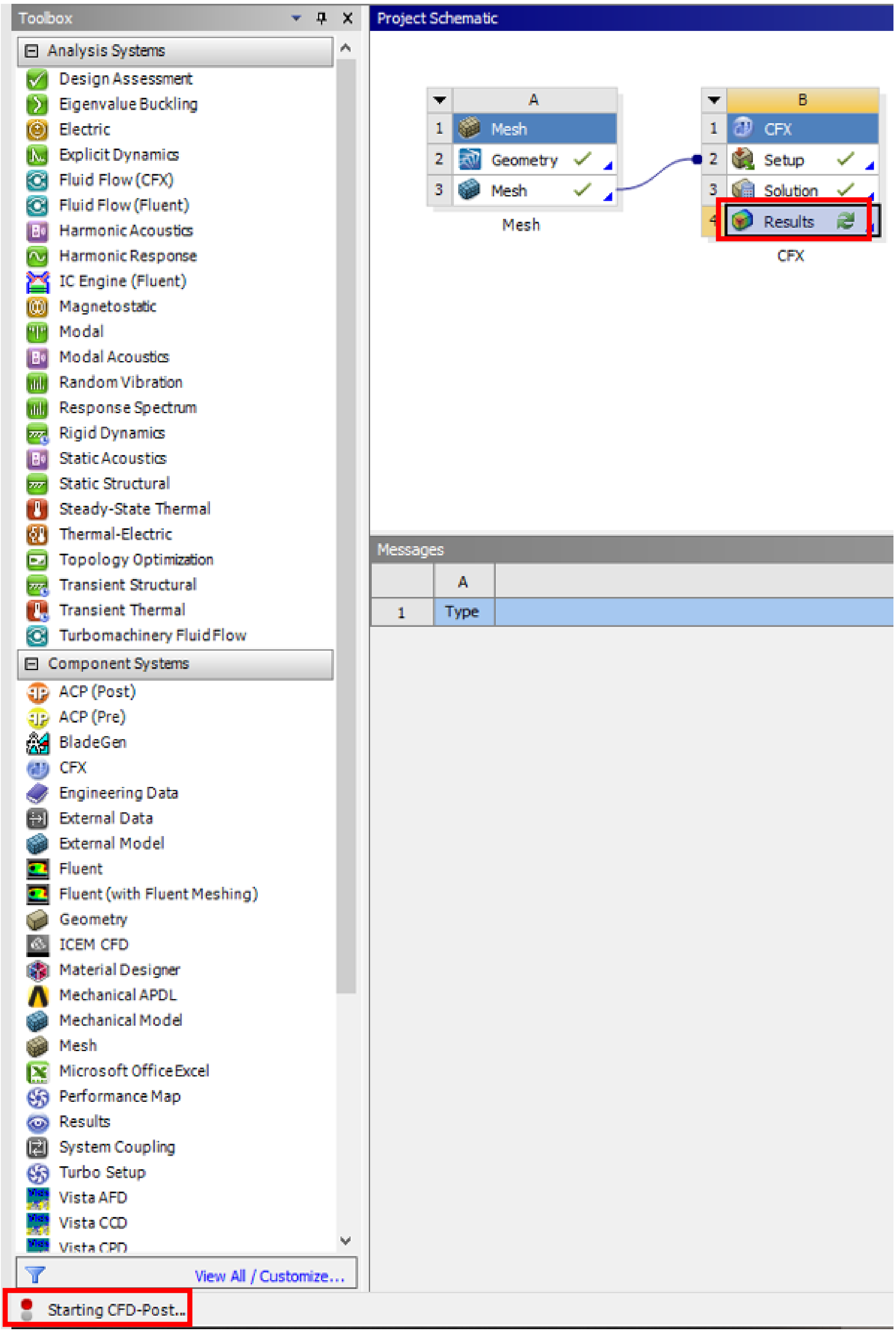
where $\lambda_1 = \lambda(T_1)$, r is pipe radius, r_1 inner pipe radius, q_L linear heat flux given by (3)

$$\dot{q}_L=\frac{T_1-T_2}{\dfrac{\ln\dfrac{r_2}{r_1}}{2\pi\lambda_m}} \tag{3}$$

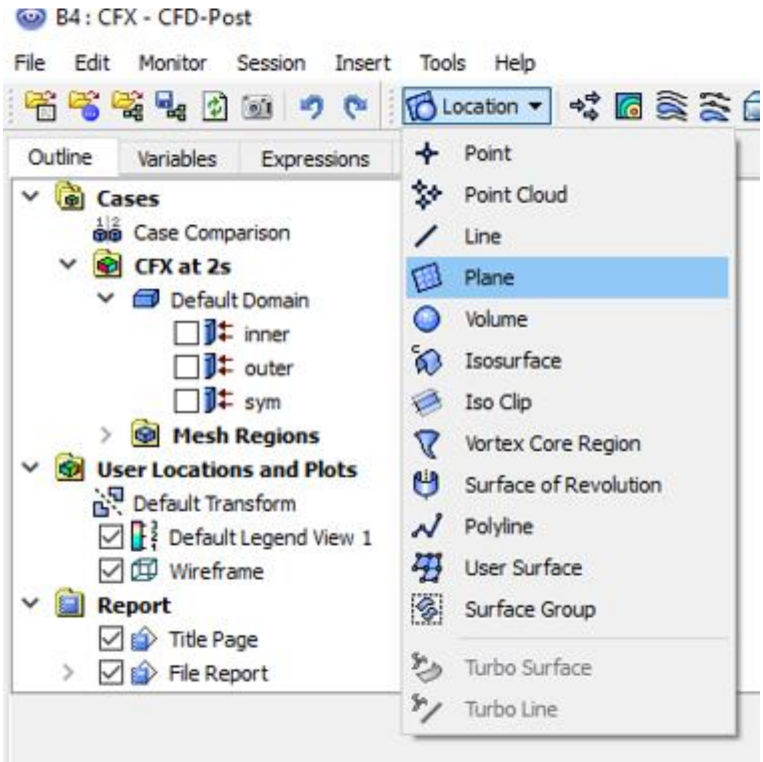
where λ_m is the average value of the thermal conductivity coefficient of the pipe material.

$$\lambda_m=\frac{1}{2}\left[\lambda(T_1)+\lambda(T_2)\right]=\frac{1}{2}(\lambda_1+\lambda_2) \tag{4}$$

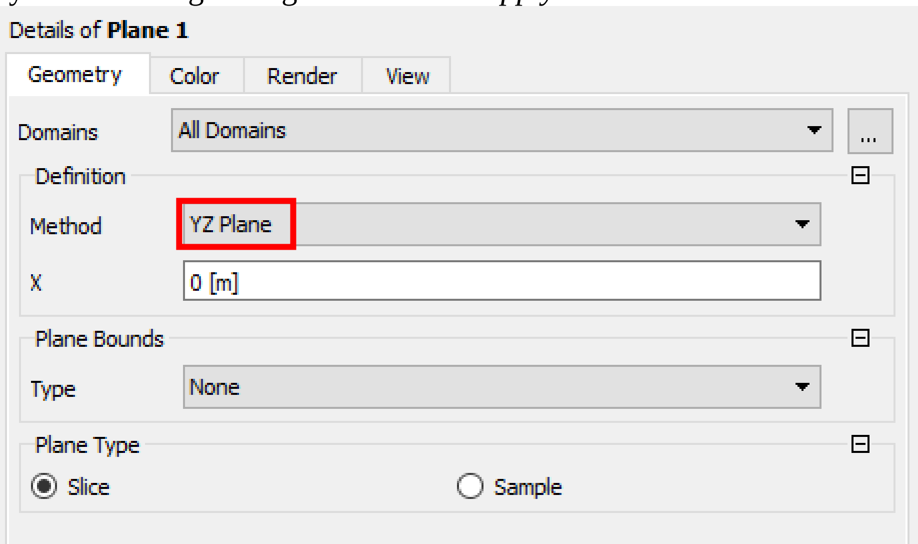
Double click LMB on *Results* to run *Ansys CFD Post* and see the results.



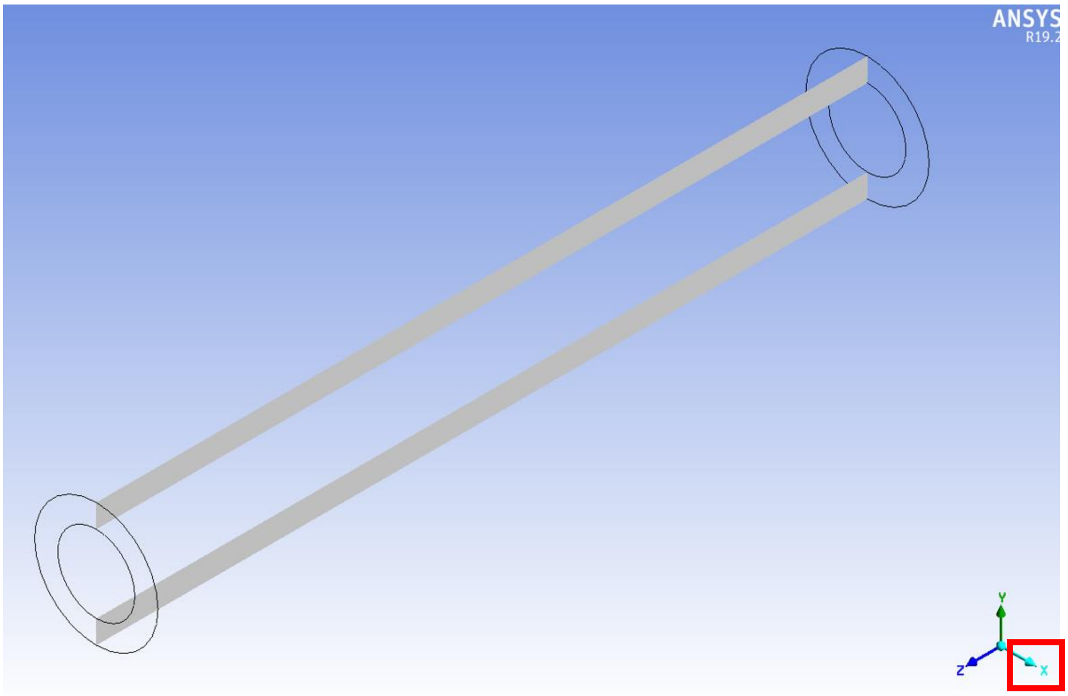
From menu *Location* select *Plane*



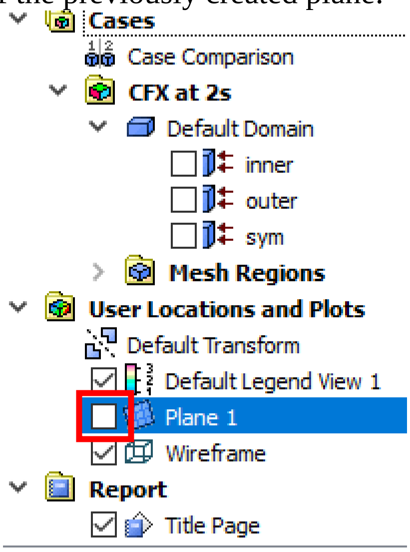
Apply the following settings and confirm *Apply*.



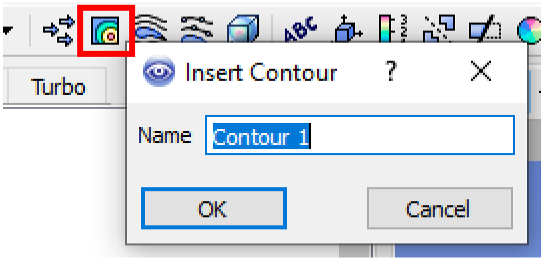
Click LMB on X axis



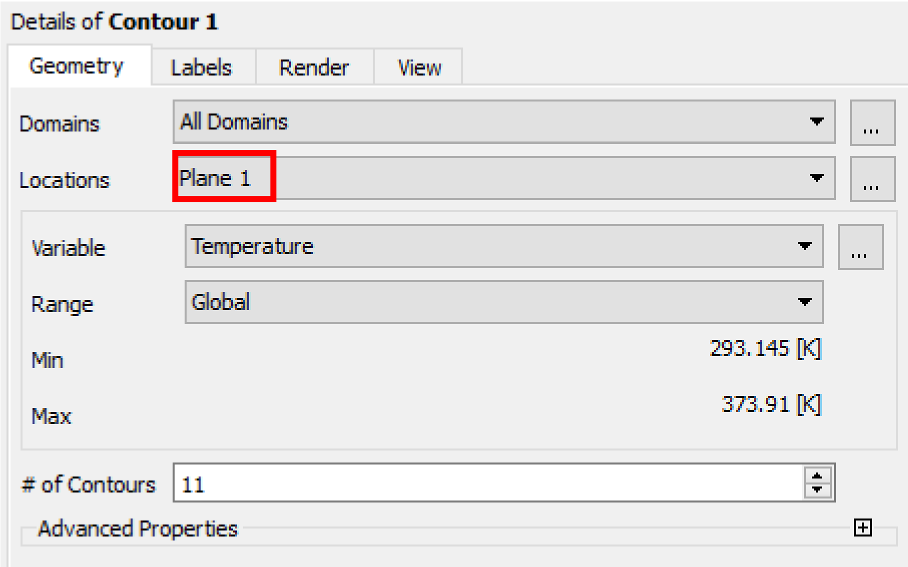
Uncheck the visibility of the previously created plane.



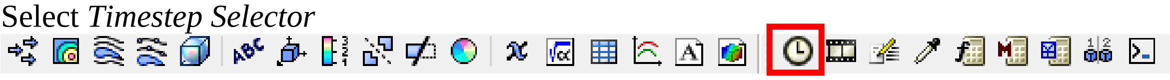
Choose contour creation and confirm *OK*.



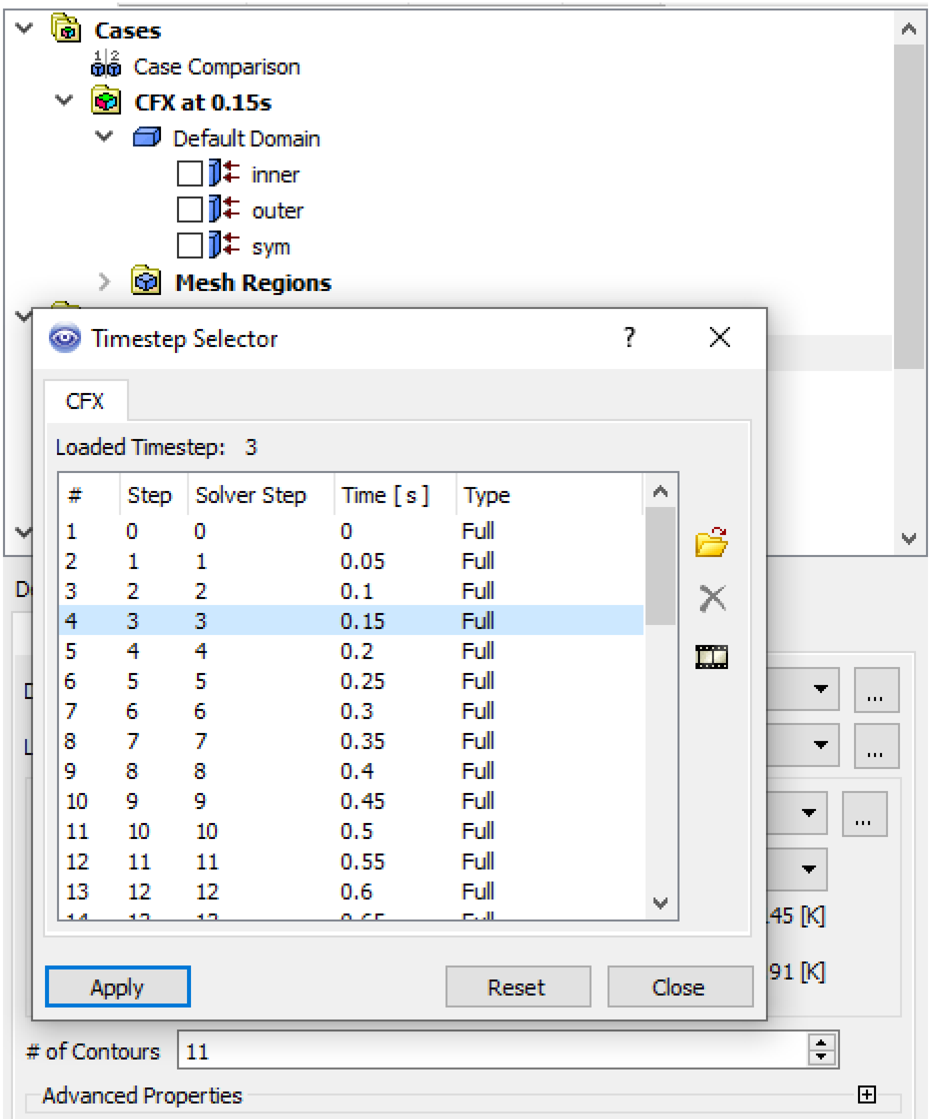
Apply the following settings and confirm *Apply*.

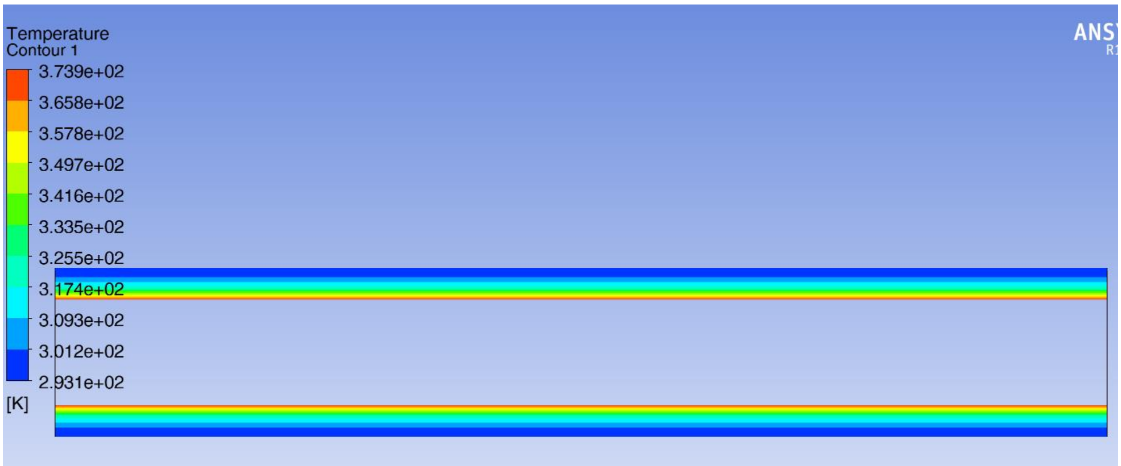


Select *Timestep Selector*

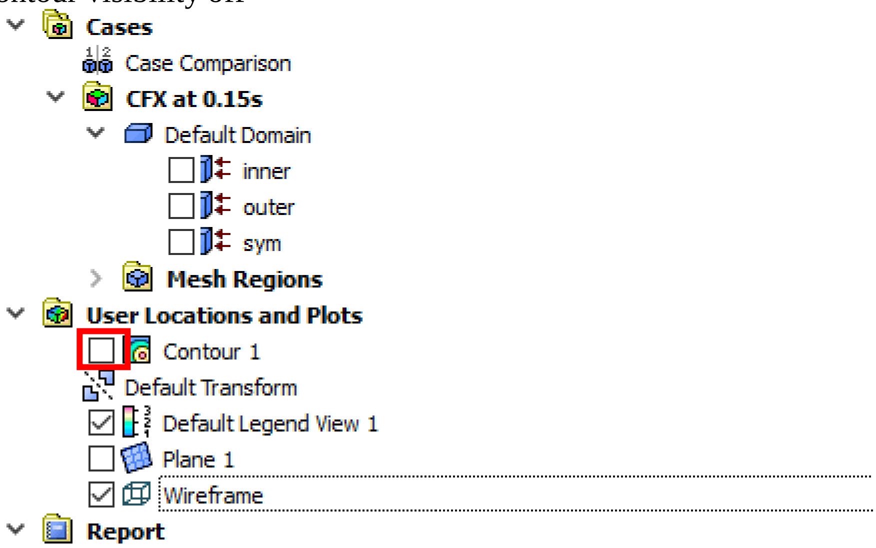


and by changing the simulation time, observe the temperature changes over time

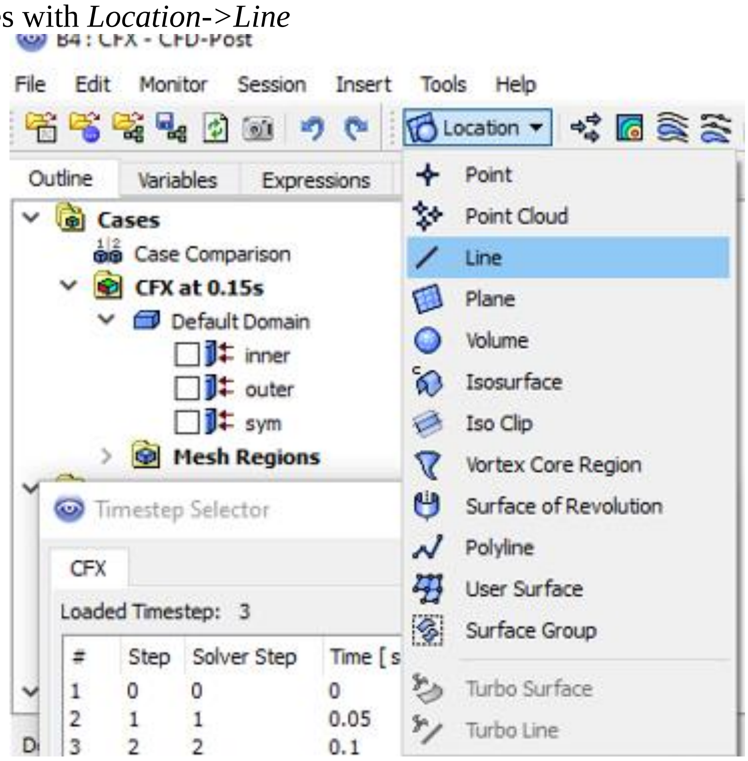




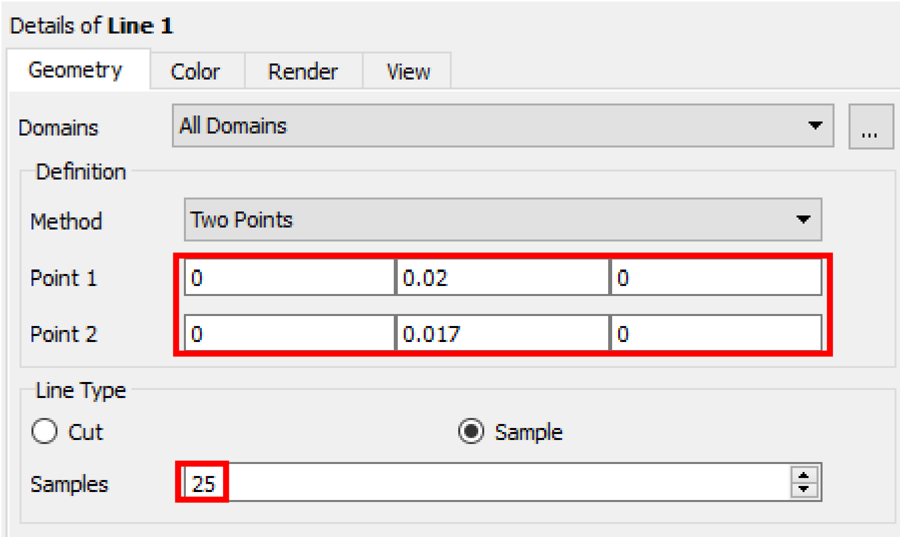
1) After saving contour pictures for the times 0; 0,1; 0,2; 0,3; 0,4; 0,5; 1,0; 2,0 s. turn contour visibility off



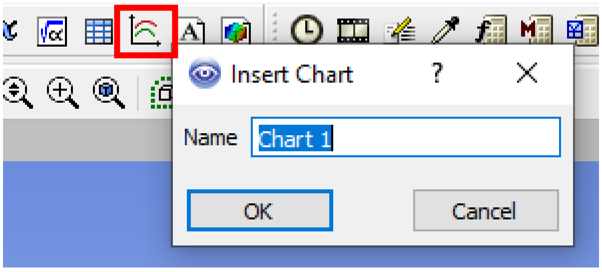
Create lines with *Location->Line*



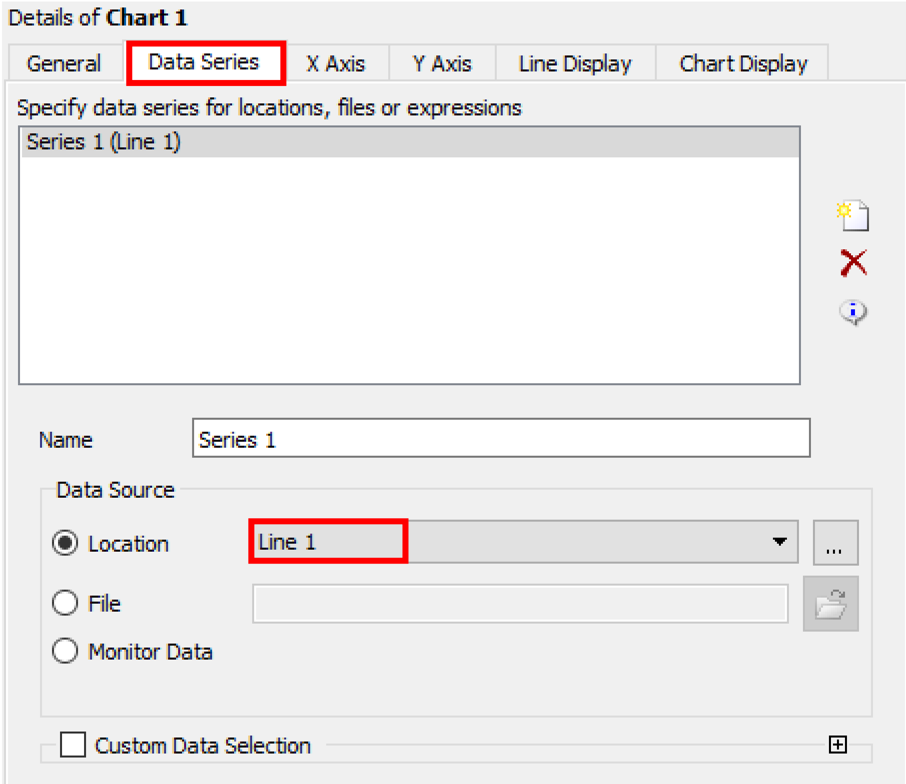
Apply the following settings and confirm *Apply*.

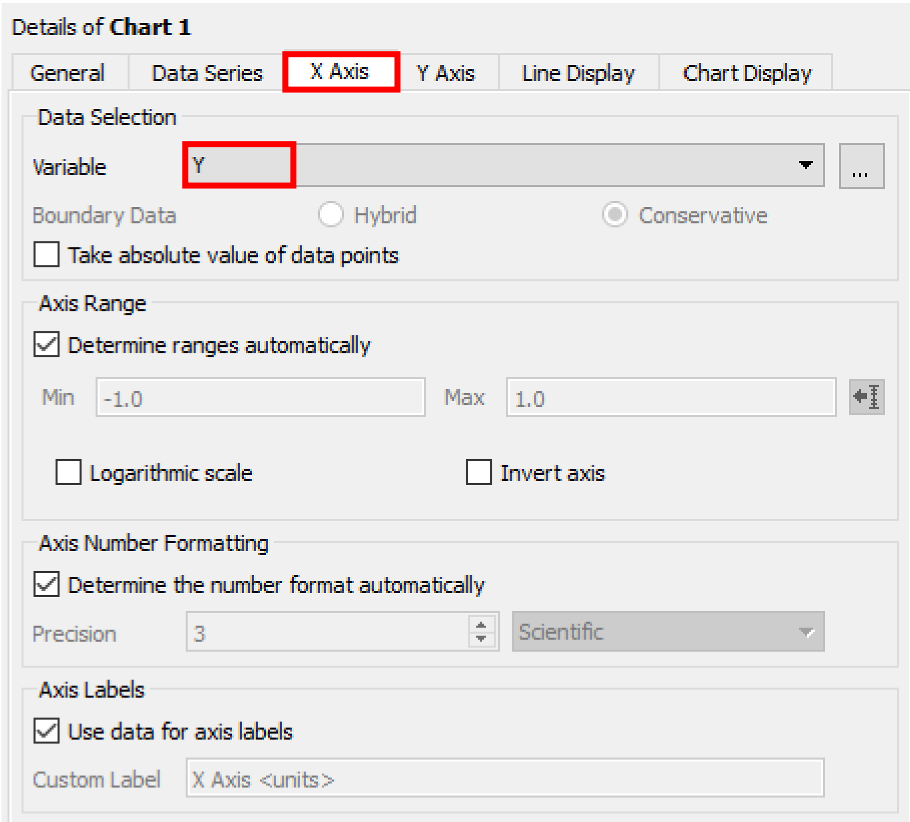


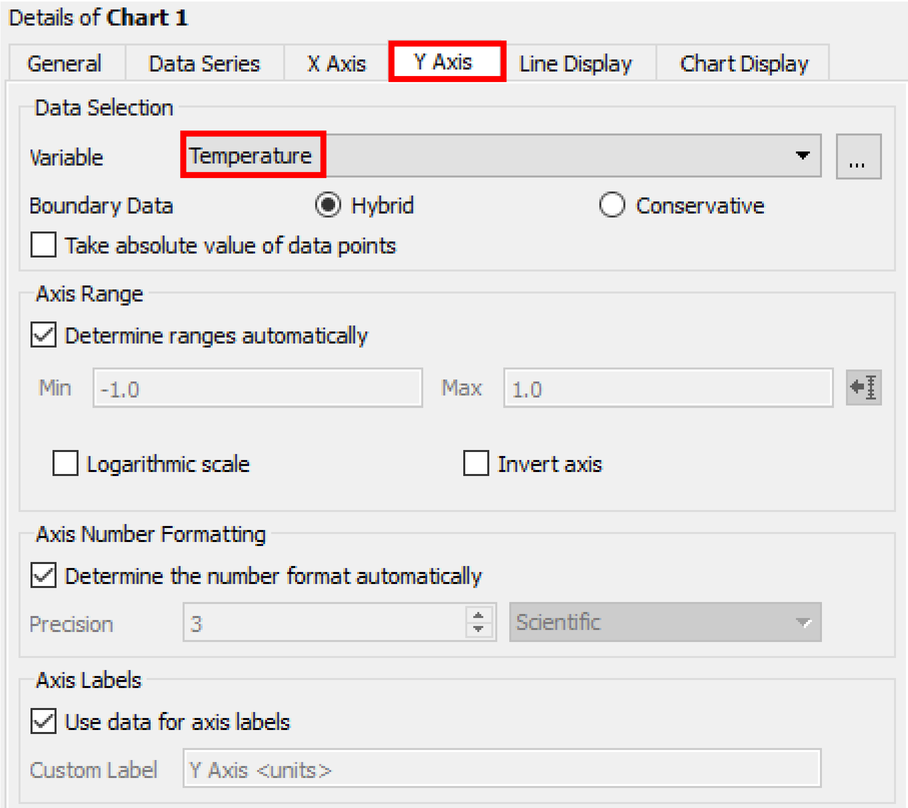
Select the chart creation icon and confirm *OK*.



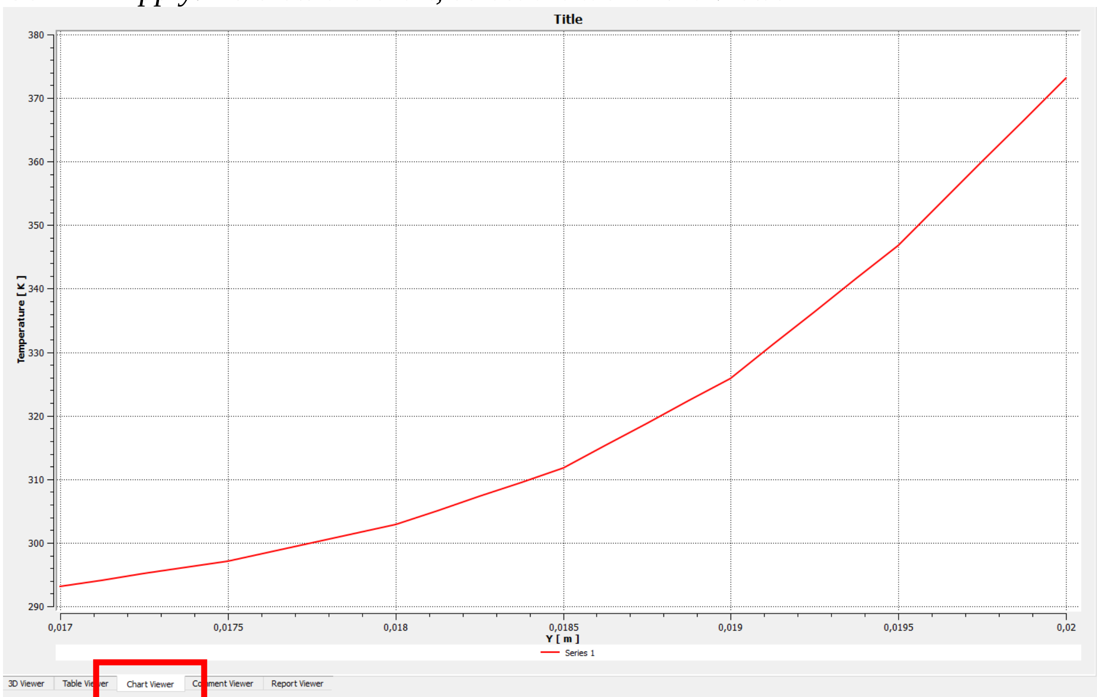
Apply the settings as in the figures below


Details of **Chart 1**
General
Data Series
X Axis
Y Axis
Line Display
Chart Display
Specify data series for locations, files or expressions
Series 1 (Line 1)
Name
Series 1
Data Source
Location
Line 1
...
File
Monitor Data
Custom Data Selection

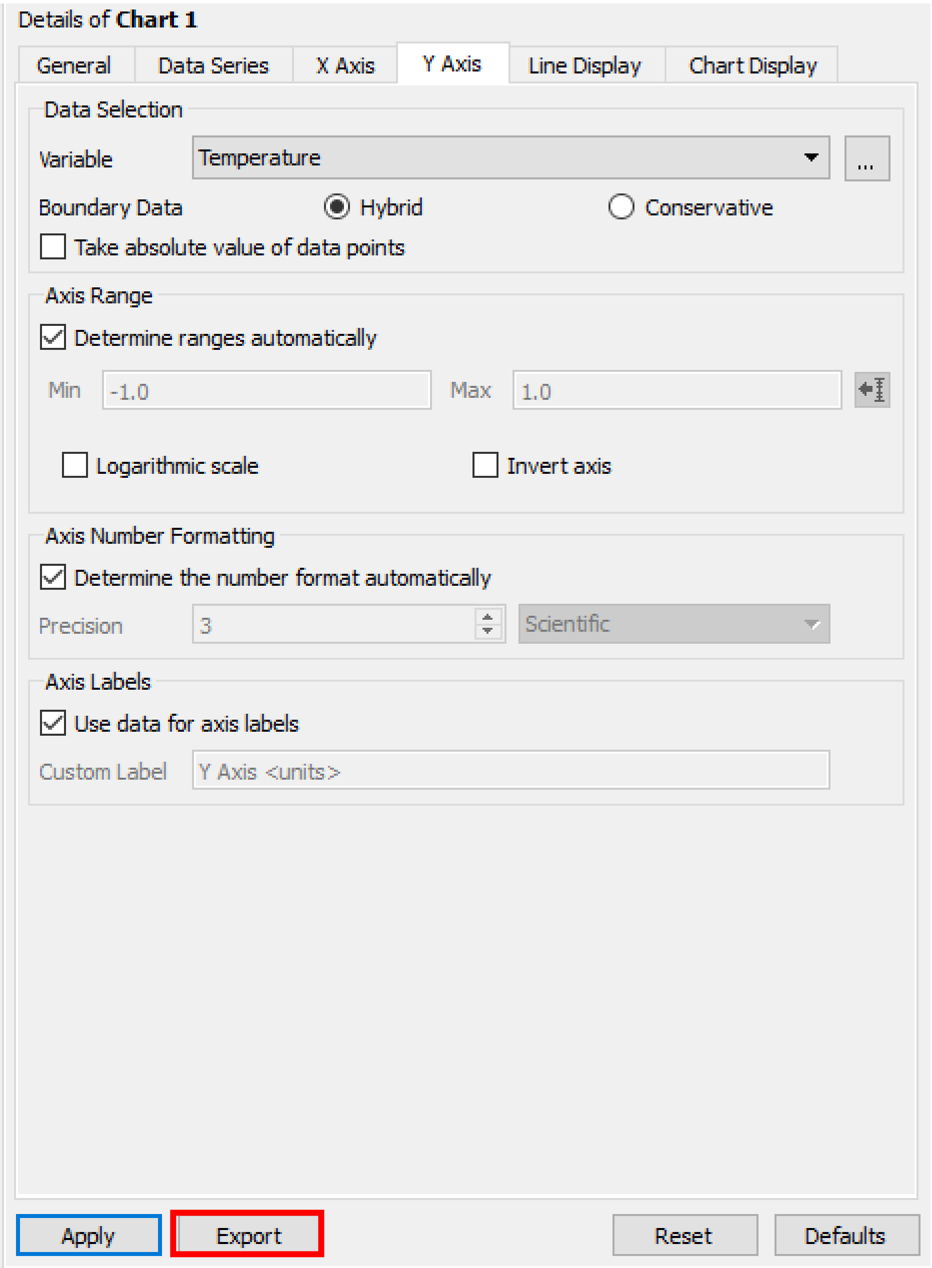

Details of **Chart 1**
General
Data Series
X Axis
Y Axis
Line Display
Chart Display
Data Selection
Variable
Y
...
Boundary Data
Hybrid
Conservative
Take absolute value of data points
Axis Range
Determine ranges automatically
Min
-1.0
Max
1.0
Logarithmic scale
Invert axis
Axis Number Formatting
Determine the number format automatically
Precision
3
Scientific
Axis Labels
Use data for axis labels
Custom Label
X Axis <units>



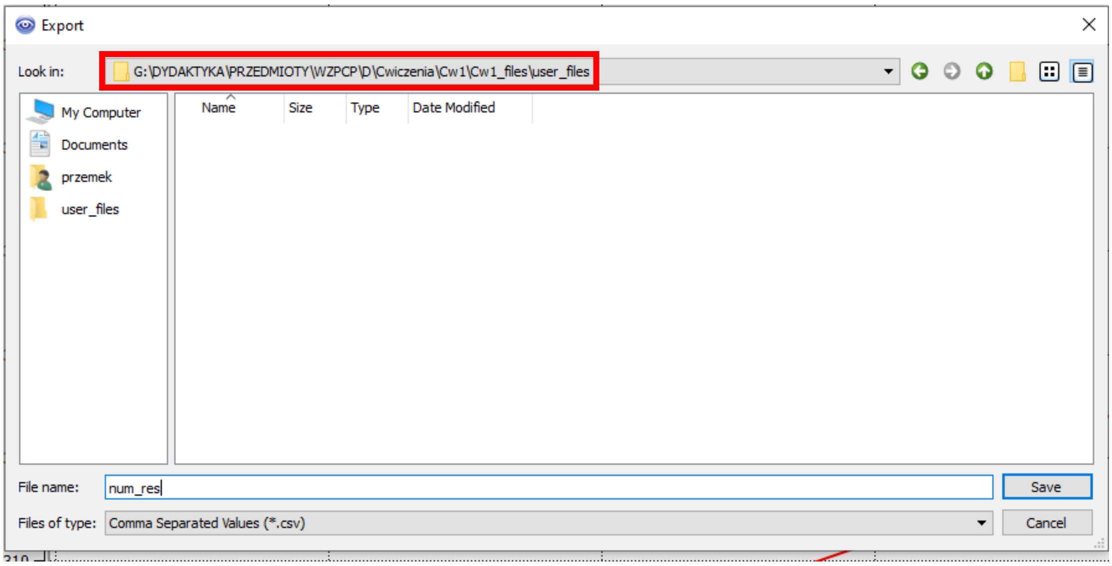
Confirm *Apply*. To view the chart, select the tab *Chart Viewer*



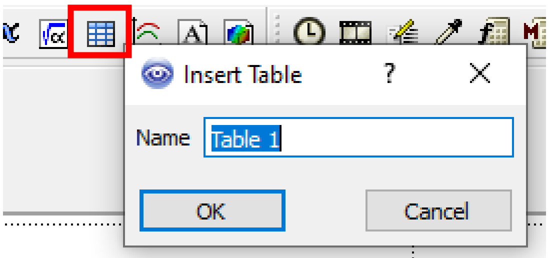
To compare numerical and analytical results, export the numerical results to a *csv* file.



Set name *num_res* and save. The location of the file is shown at the top of the dialog box and is different on each computer.



8) Calculate the heat transfer through the pipe
Create table

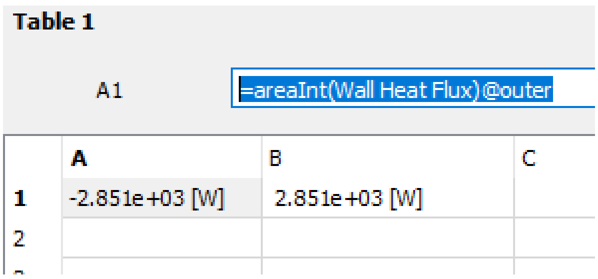


Make sure the results for t = 2 s are loaded (*Timestep Selector* icon). Into cell A1, paste the expression calculating the area integral of heat flux on the outer surface of the pipe

=areaInt(Wall Heat Flux)@outer

and press *Enter*. Into cell B1 paste the expression calculating the area integral of heat flux on the inner surface of the pipe

=areaInt(Wall Heat Flux)@inner



9) Additional task: Create an animation showing temperature changes across the pipe (help: http://fluid.itcmp.pwr.wroc.pl/~pblasiak/CFD/UsefulInformation/animationCFX.jpg)

Results to be included in the report:

2) contours of temperature distributions in a cylindrical partition for times: 0; 0,1; 0,2; 0,3; 0,4; 0,5; 1,0; 2,0 s.

3) compare on one chart the numerical solution of the wall temperature distribution for time $t = 2$ s with the analytical solution (2)
4) present in the table the heat flux calculated numerically and analytically (3)