MODELING & ANALYSIS OF FLUID FLOW IN CIRCULAR TUBE USING ANSYS FLUENT

UNILAG ANSYS HANDS-ON TUTORIAL 1C (MEG 222)

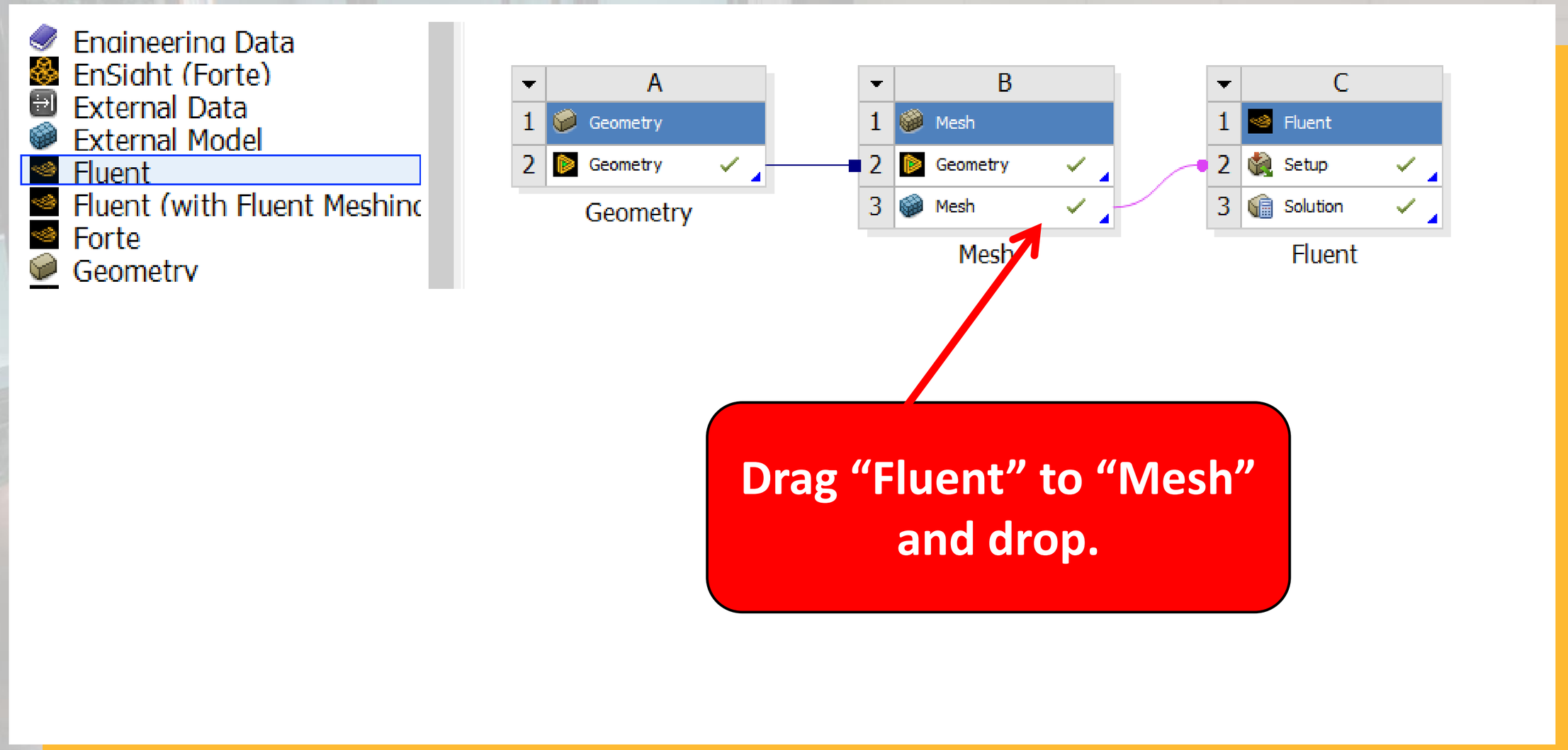
FLUID FLOW MODELING

At the end of this third tutorial, you will be able to

01 **Set up the physics of the problem in the Fluent Solver**

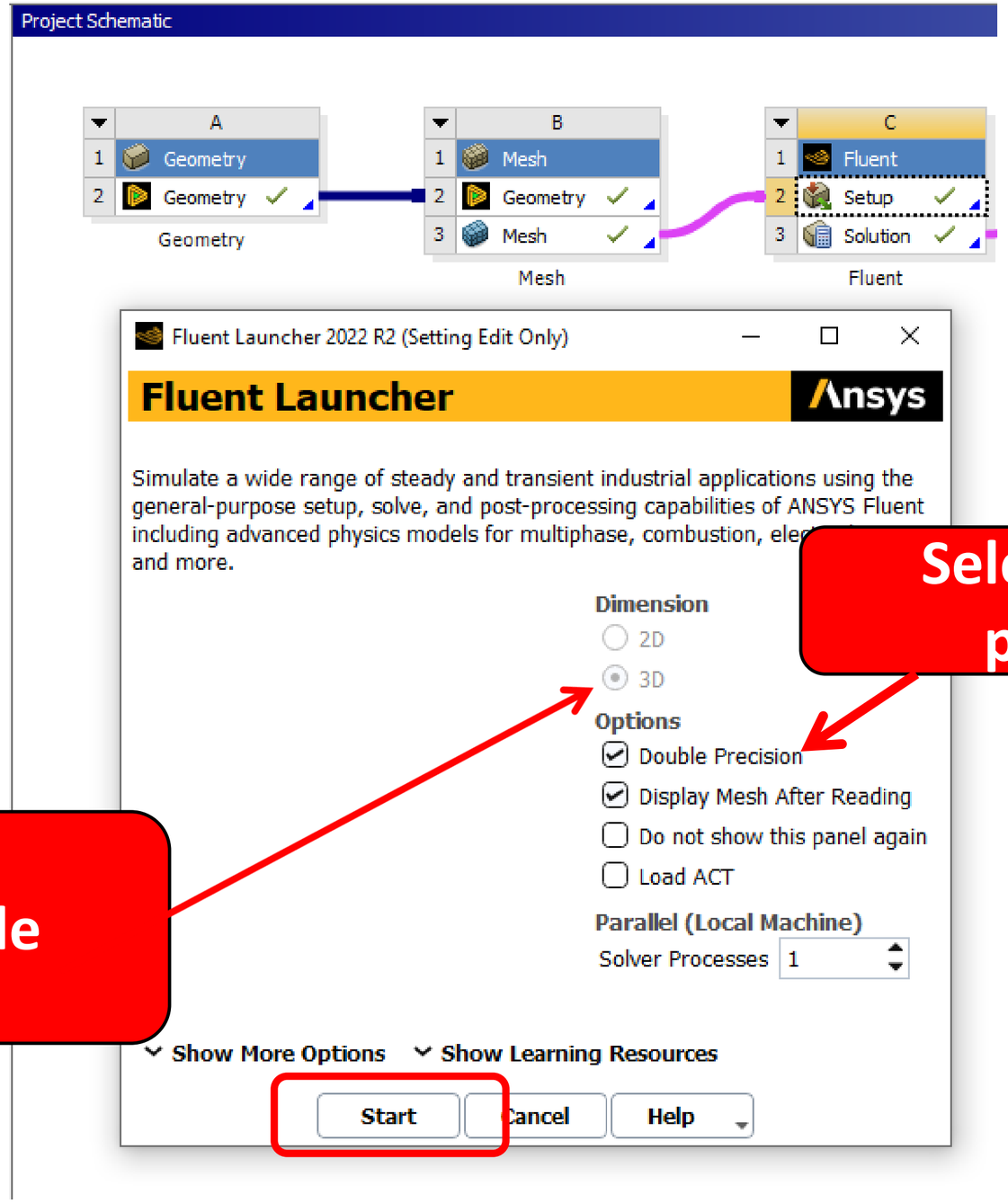
02 **Analyze the results in the post-processor**

SETUP THE PHYSICS


Engineering Data
EnSight (Forte)
External Data
External Model
Fluent
Fluent (with Fluent Meshing
Forte
Geometry
A
1 Geometry
2 Geometry
Geometry
B
1 Mesh
2 Geometry
3 Mesh
Mesh
C
1 Fluent
2 Setup
3 Solution
Fluent
Drag “Fluent” to “Mesh” and drop.

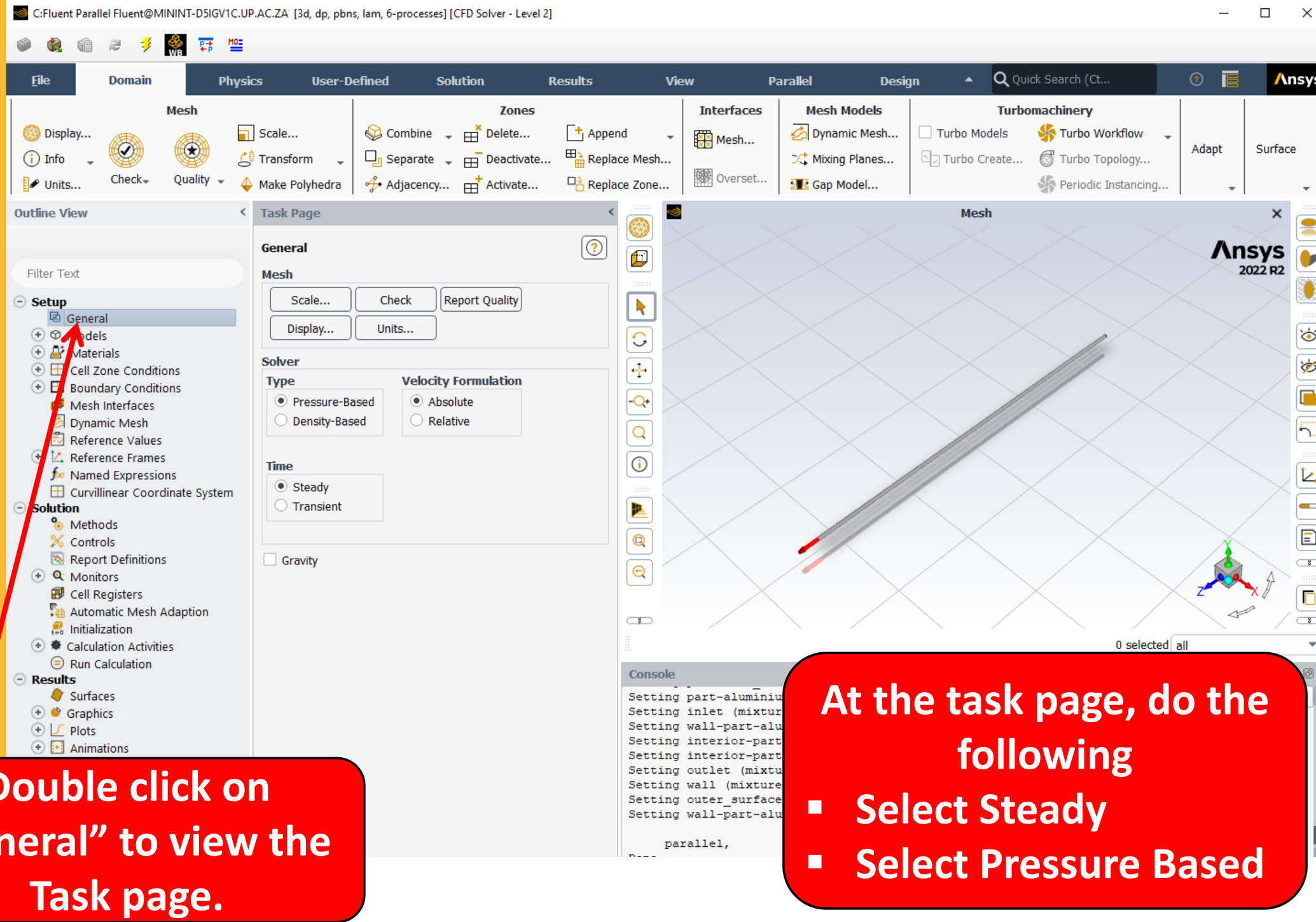
PHYSICS

SETUP



PHYSICS

SETUP



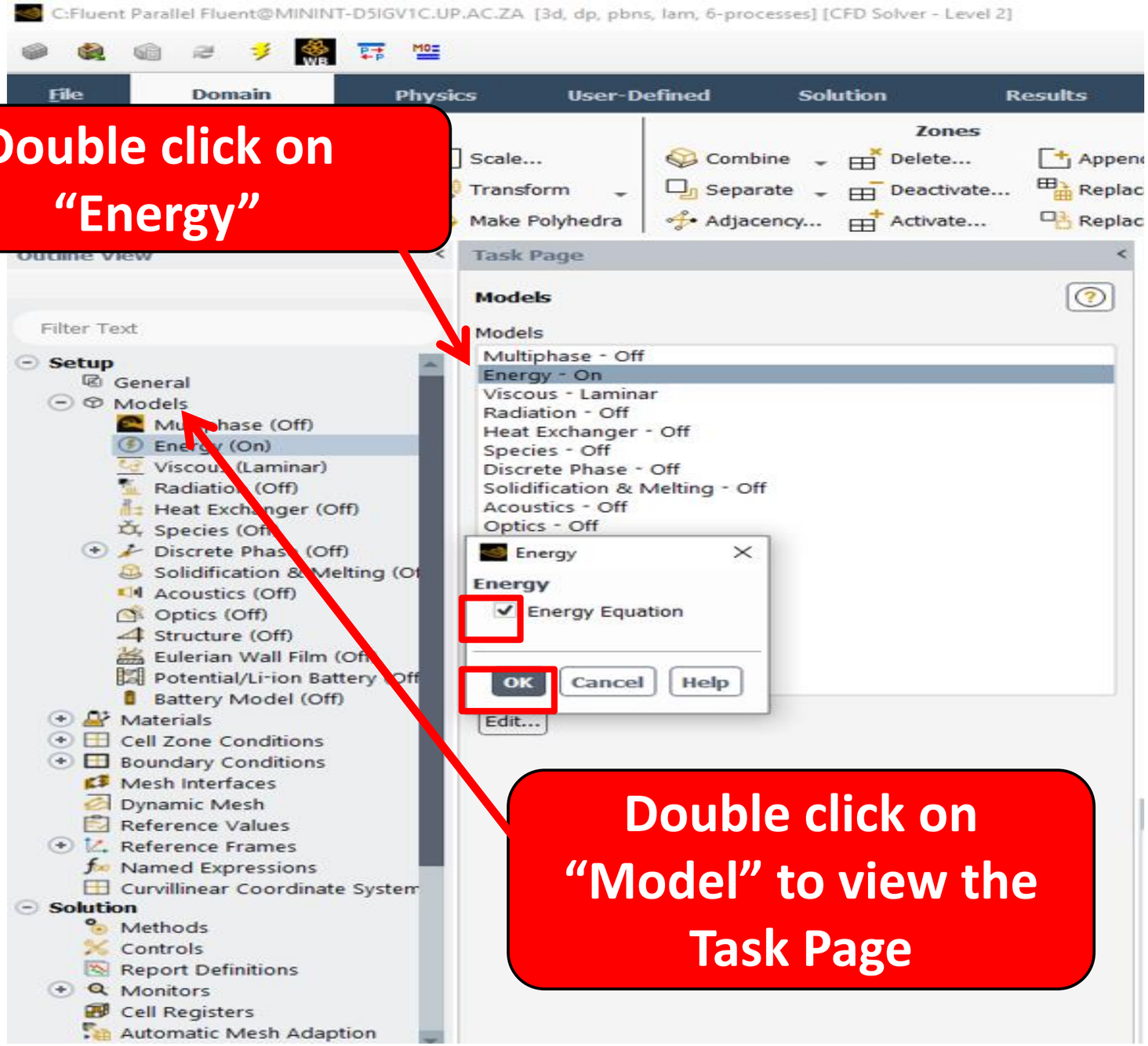
Double click on "General" to view the Task page.

At the task page, do the following

- Select Steady
- Select Pressure Based

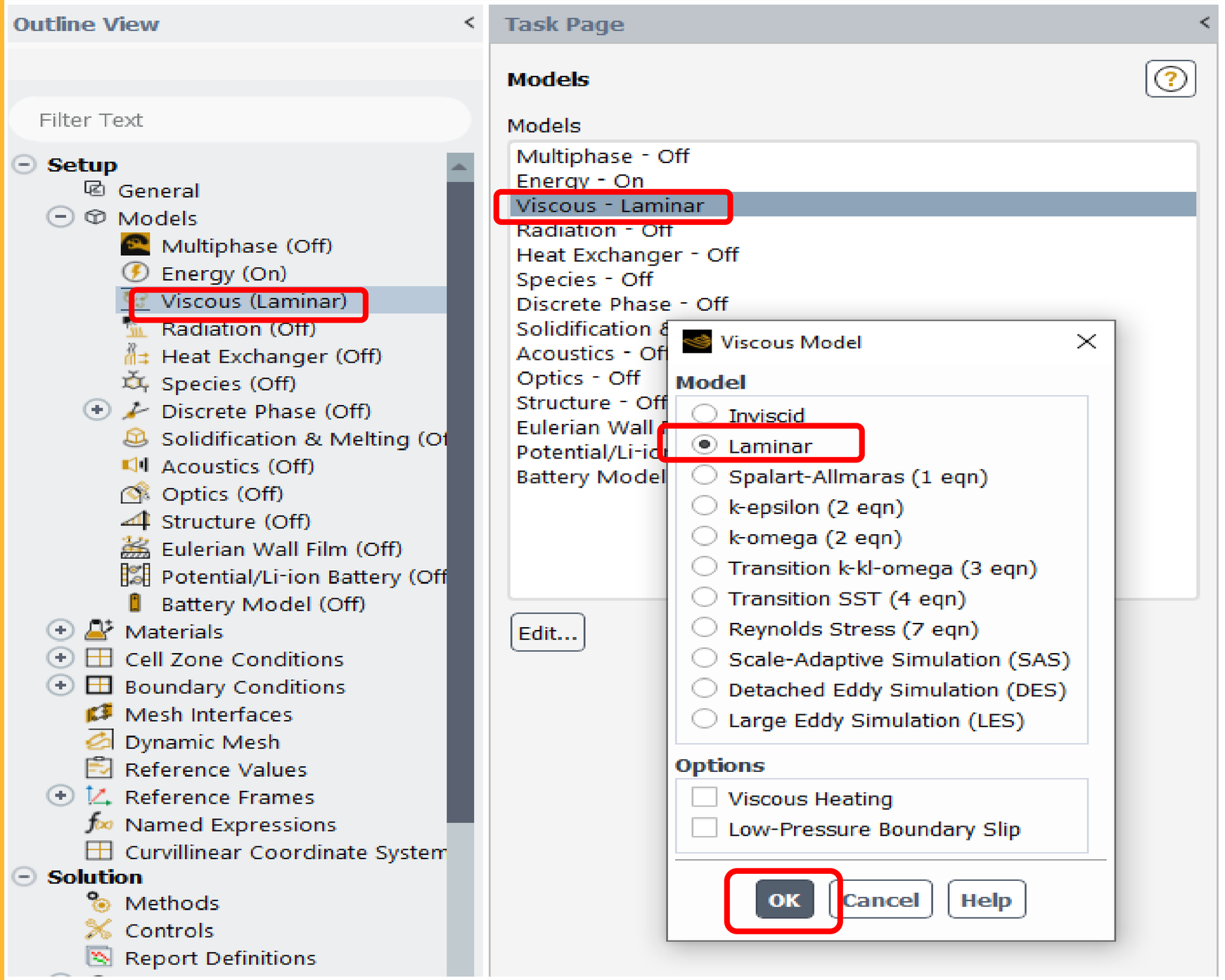
PHYSICS

SETUP



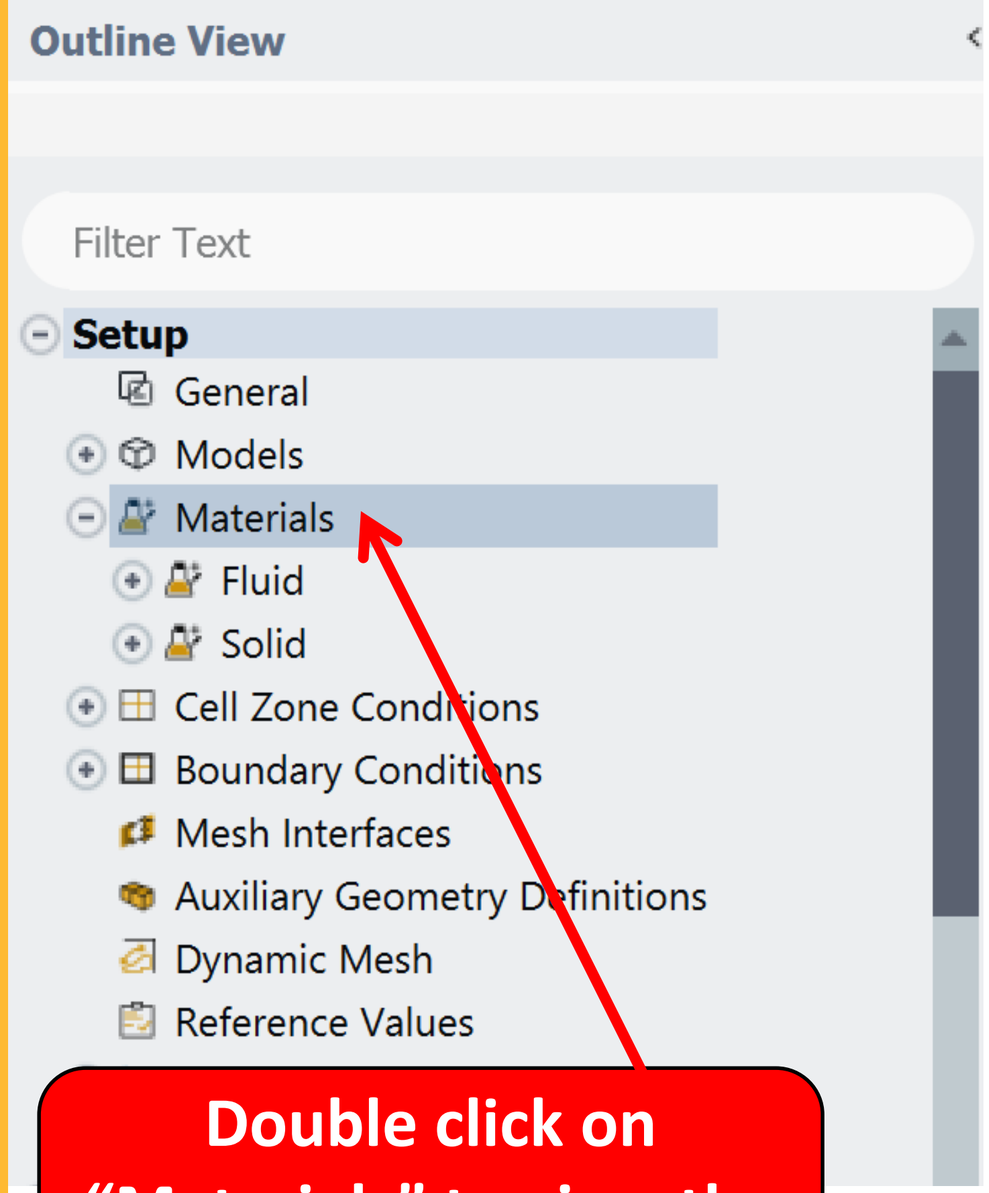
PHYSICS

SETUP

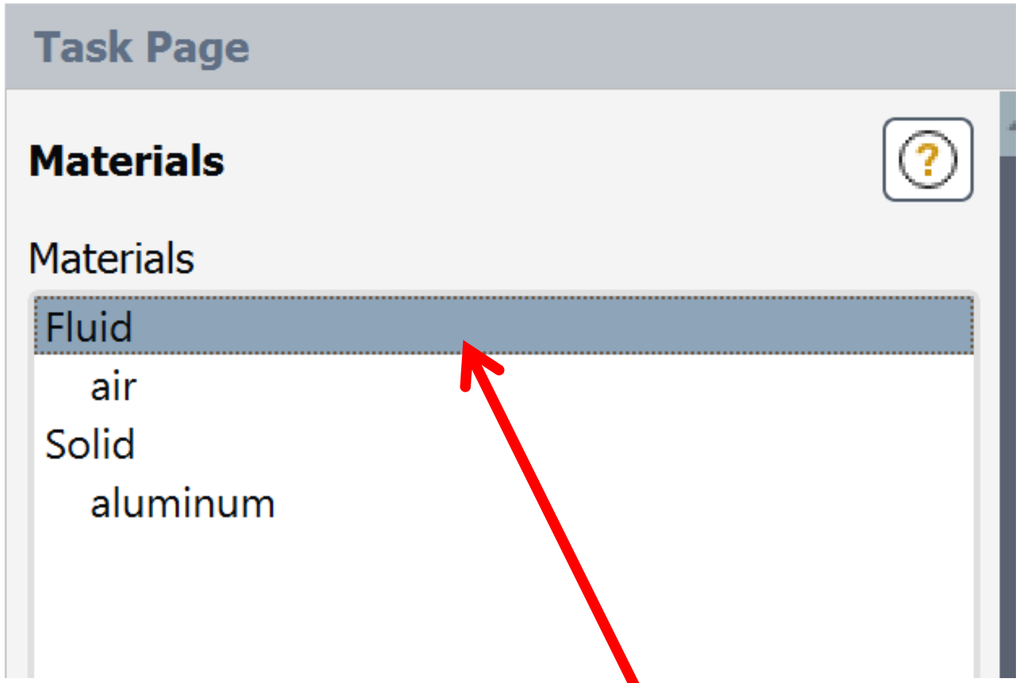


PHYSICS

SETUP



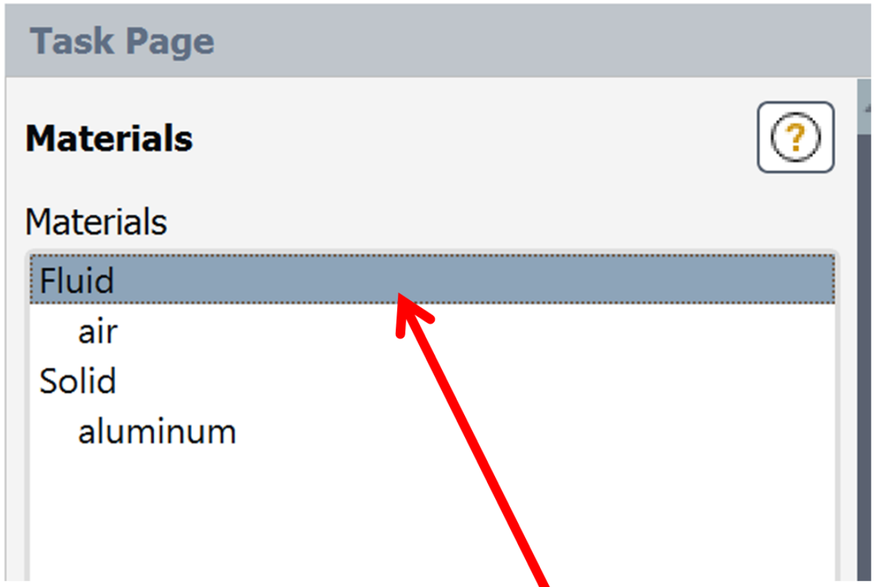
Double click on “Materials” to view the Task page



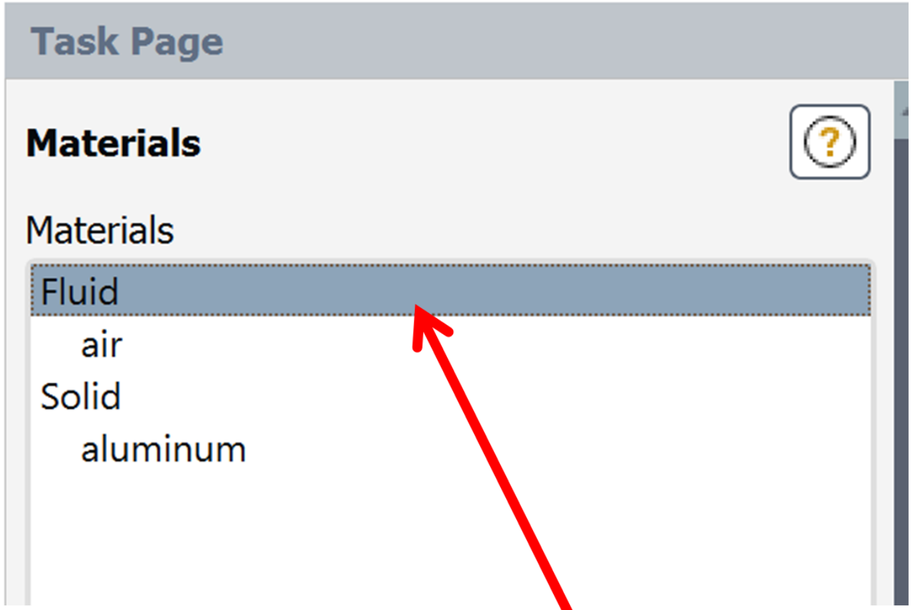
Double click on “Fluid” to add the required fluid from Fluent Database.

PHYSICS

SETUP



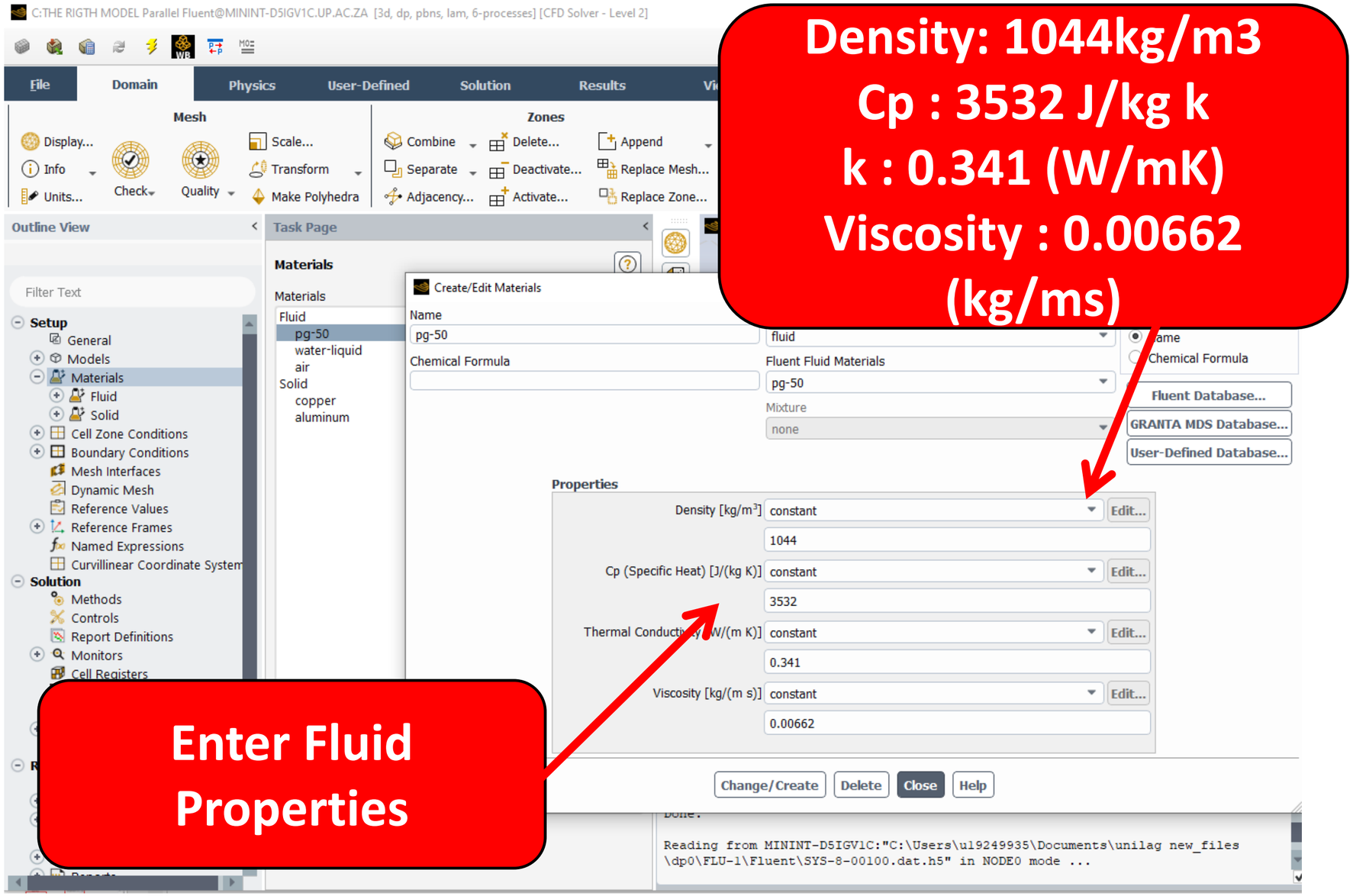
Double click on “Fluid” to add the required fluid properties (Mixture of water and 30% PG).



Double click on “Fluid” to add the required fluid properties (Mixture of water and 50% PG).

PHYSICS

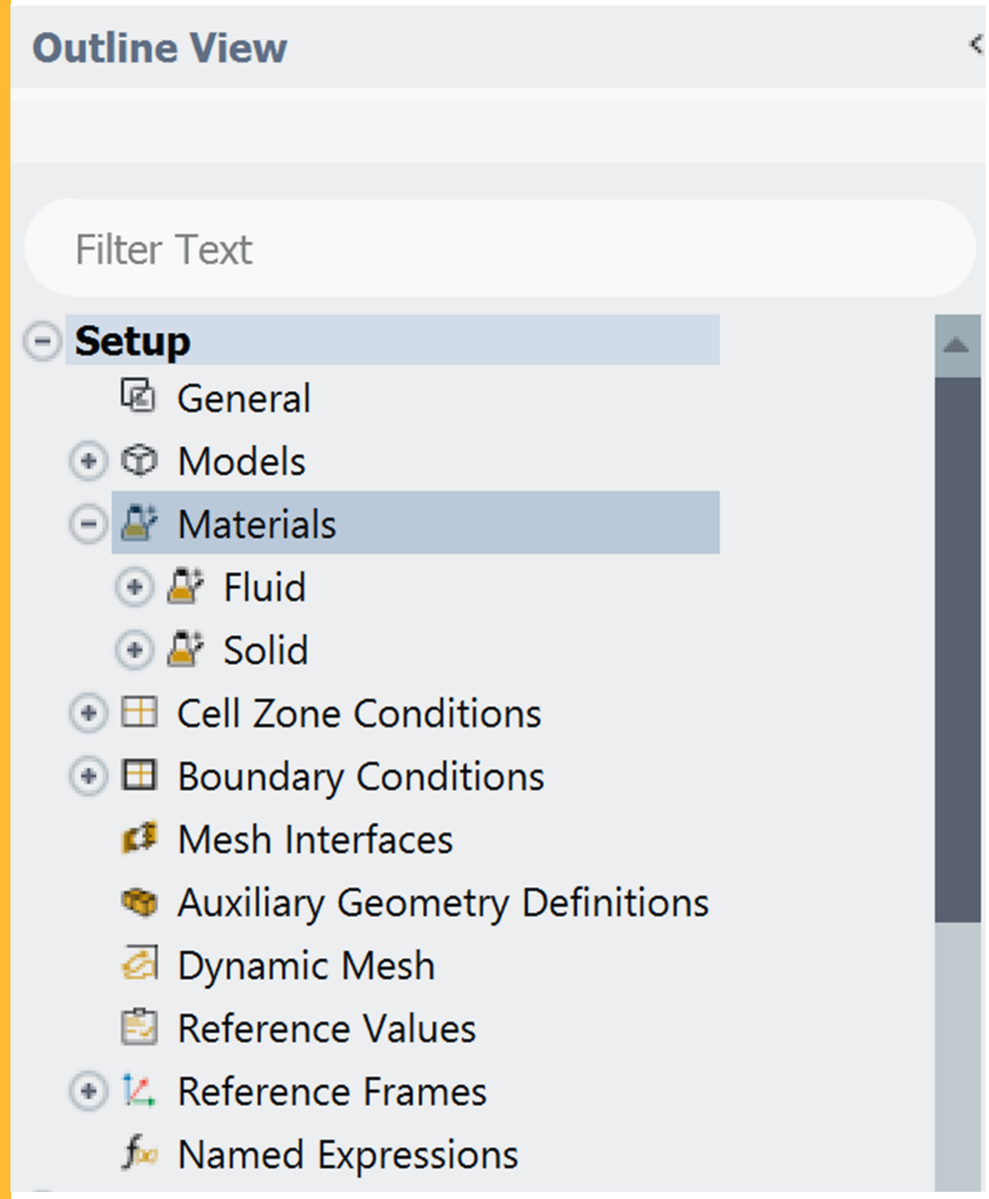
SETUP

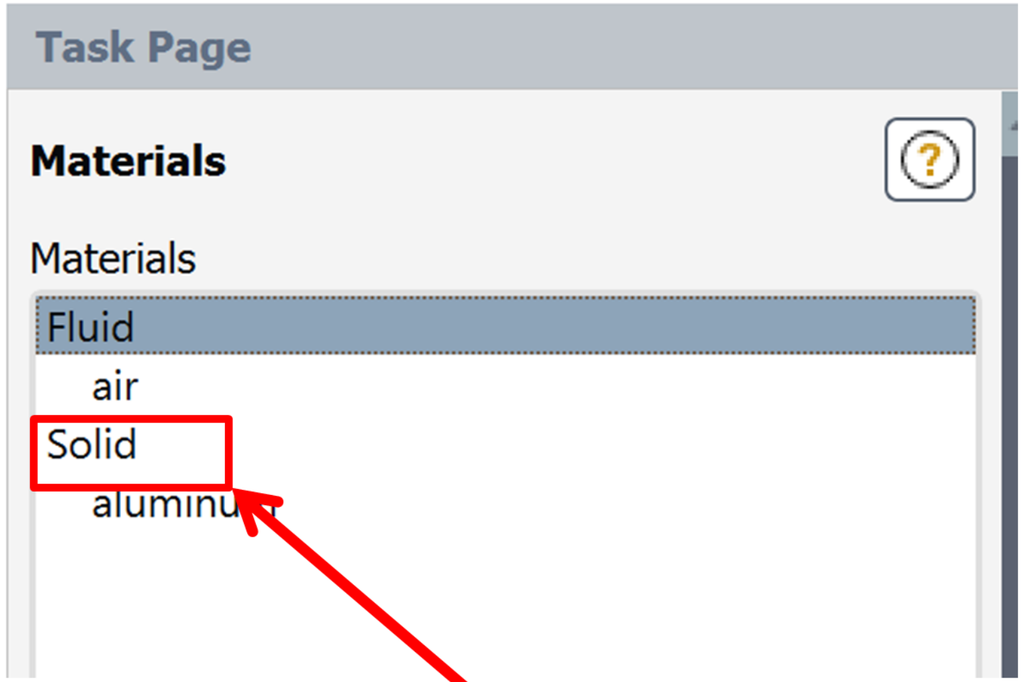


This is an example showing you how to enter the fluid properties for water + 50% ethylene glycol (pg-50). Do the same for pure water and for water + 30% ethylene glycol (pg-30)

PHYSICS

SETUP

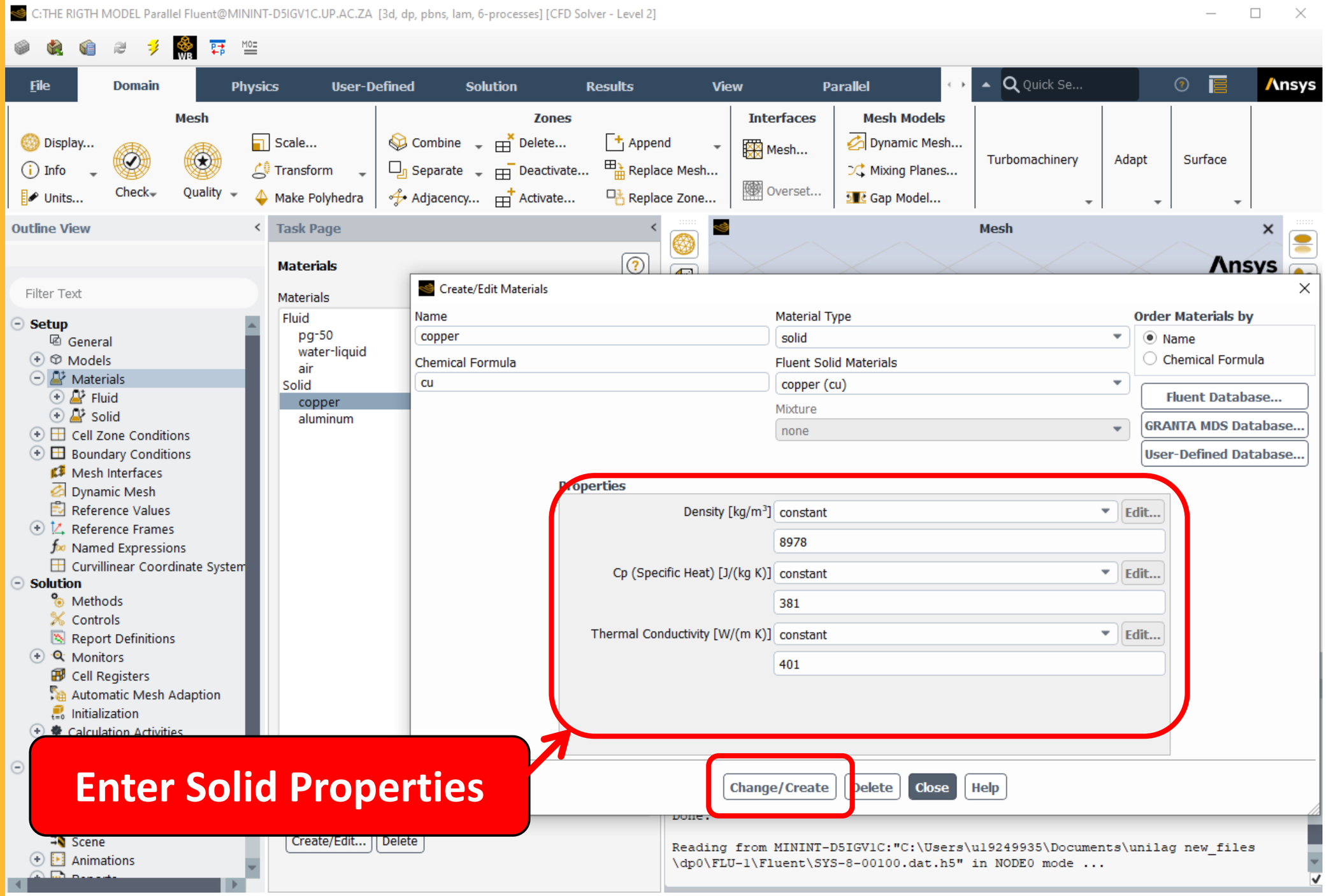




Double click on “Solid” and add the required solid properties (Copper).

PHYSICS

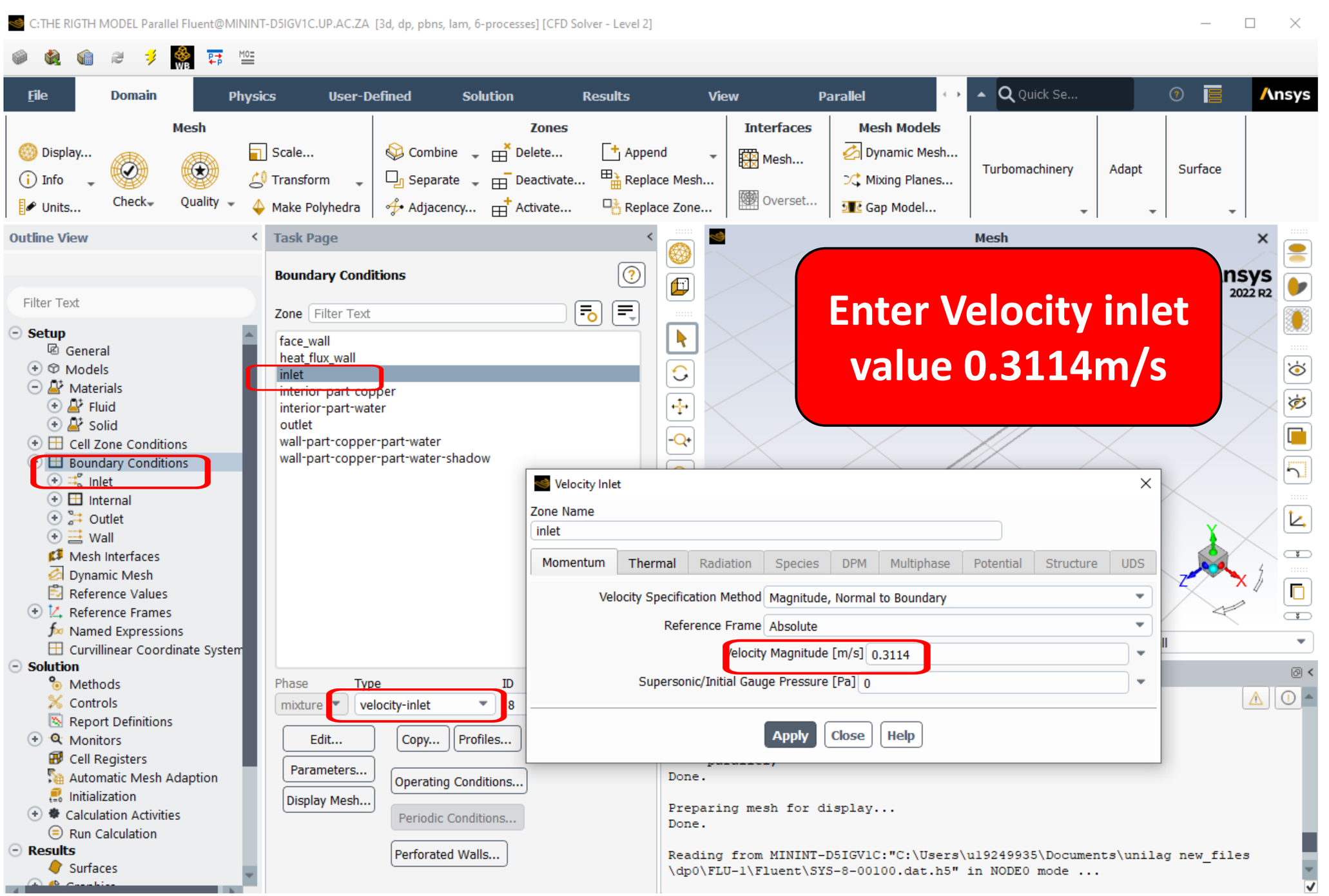
SETUP



Enter Solid Properties

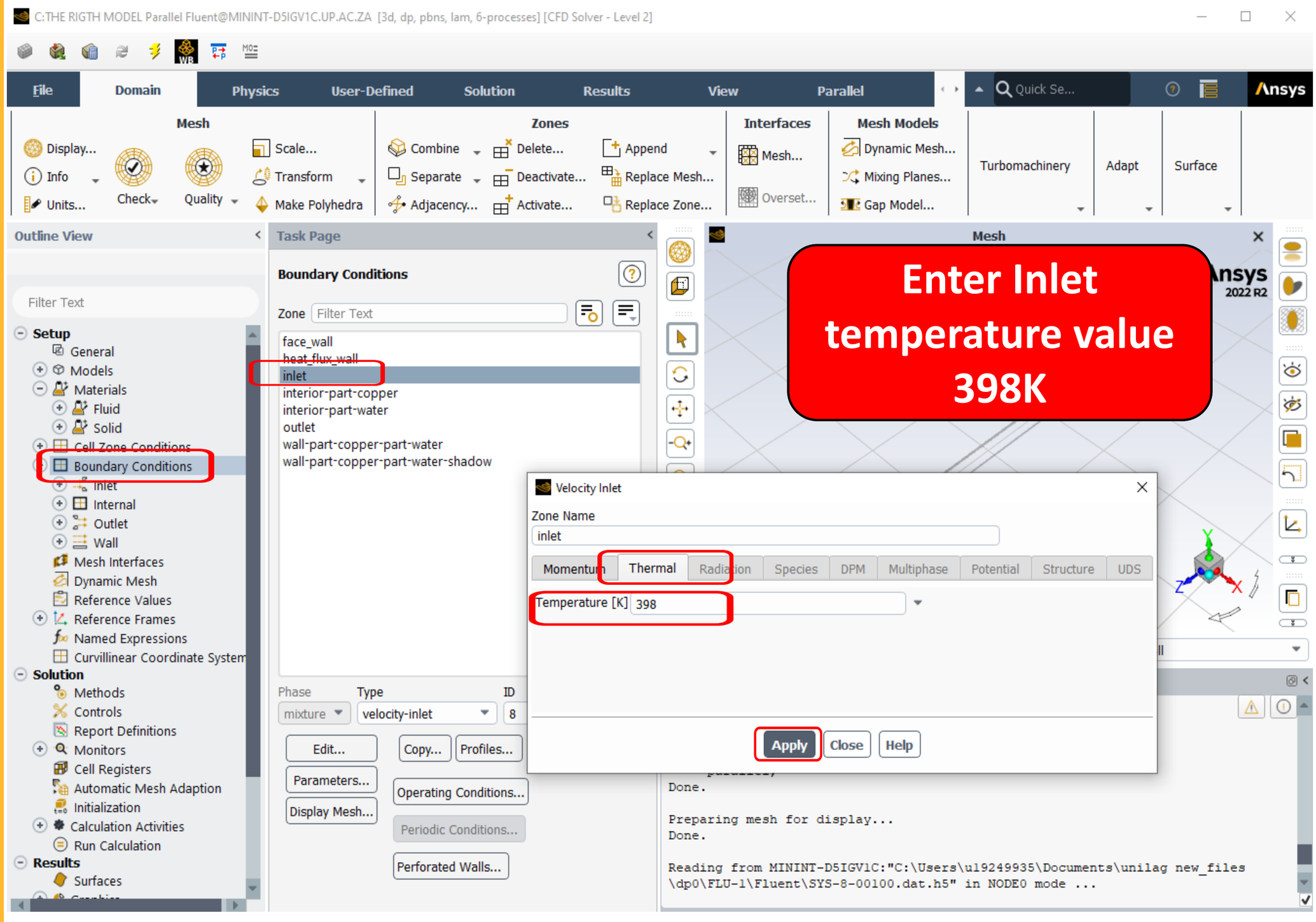
PHYSICS

SETUP



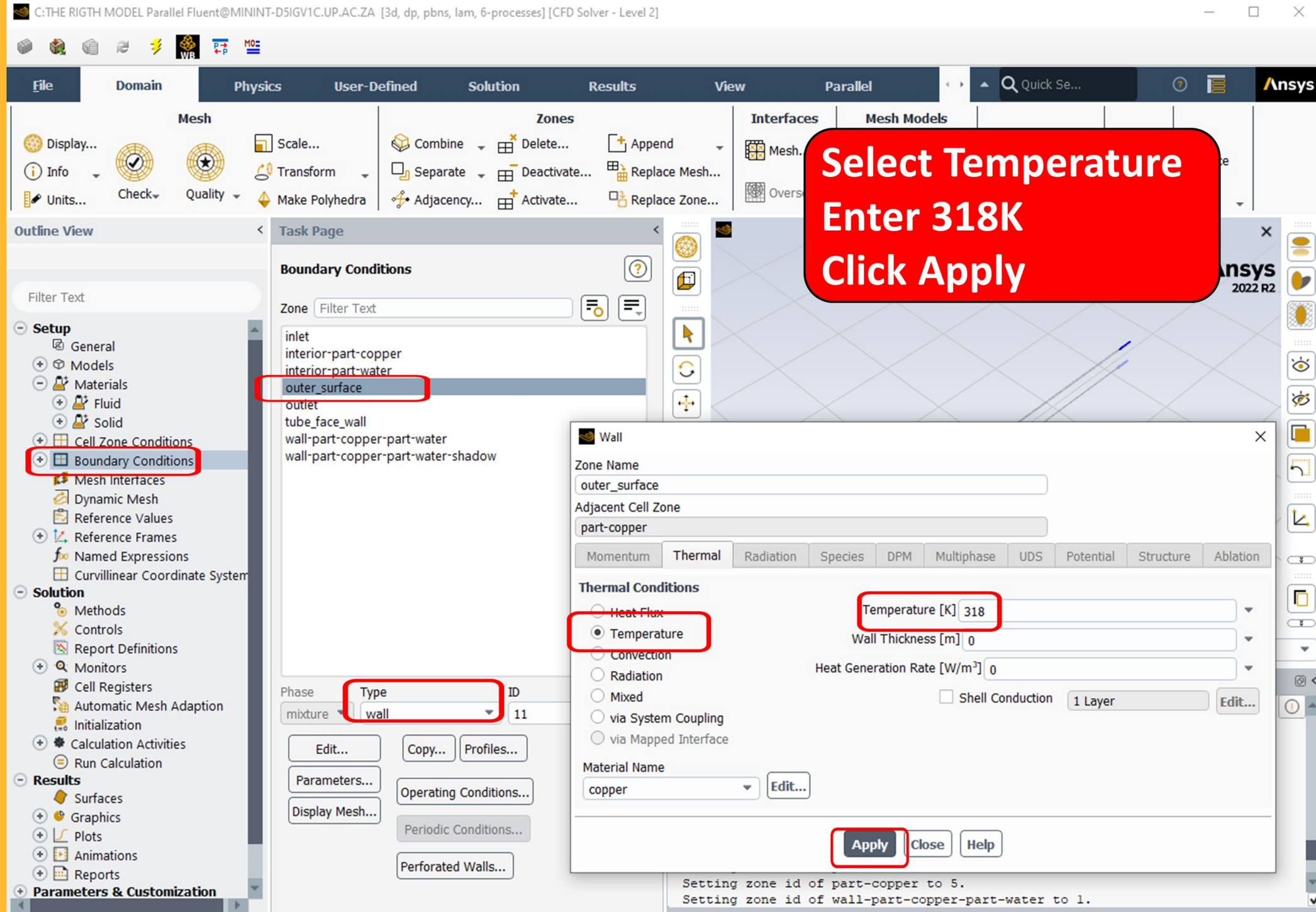
PHYSICS

SETUP



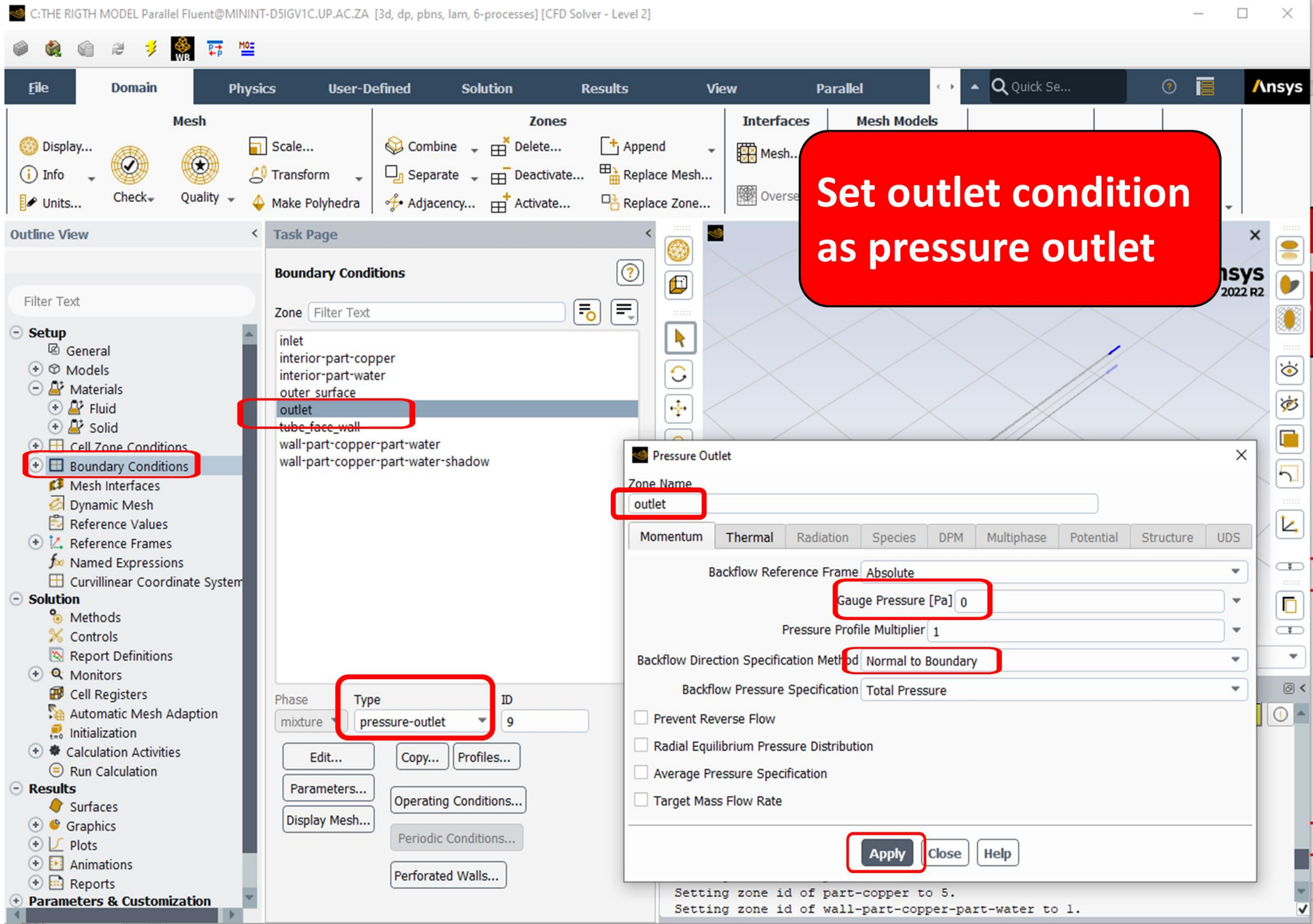
PHYSICS

SETUP



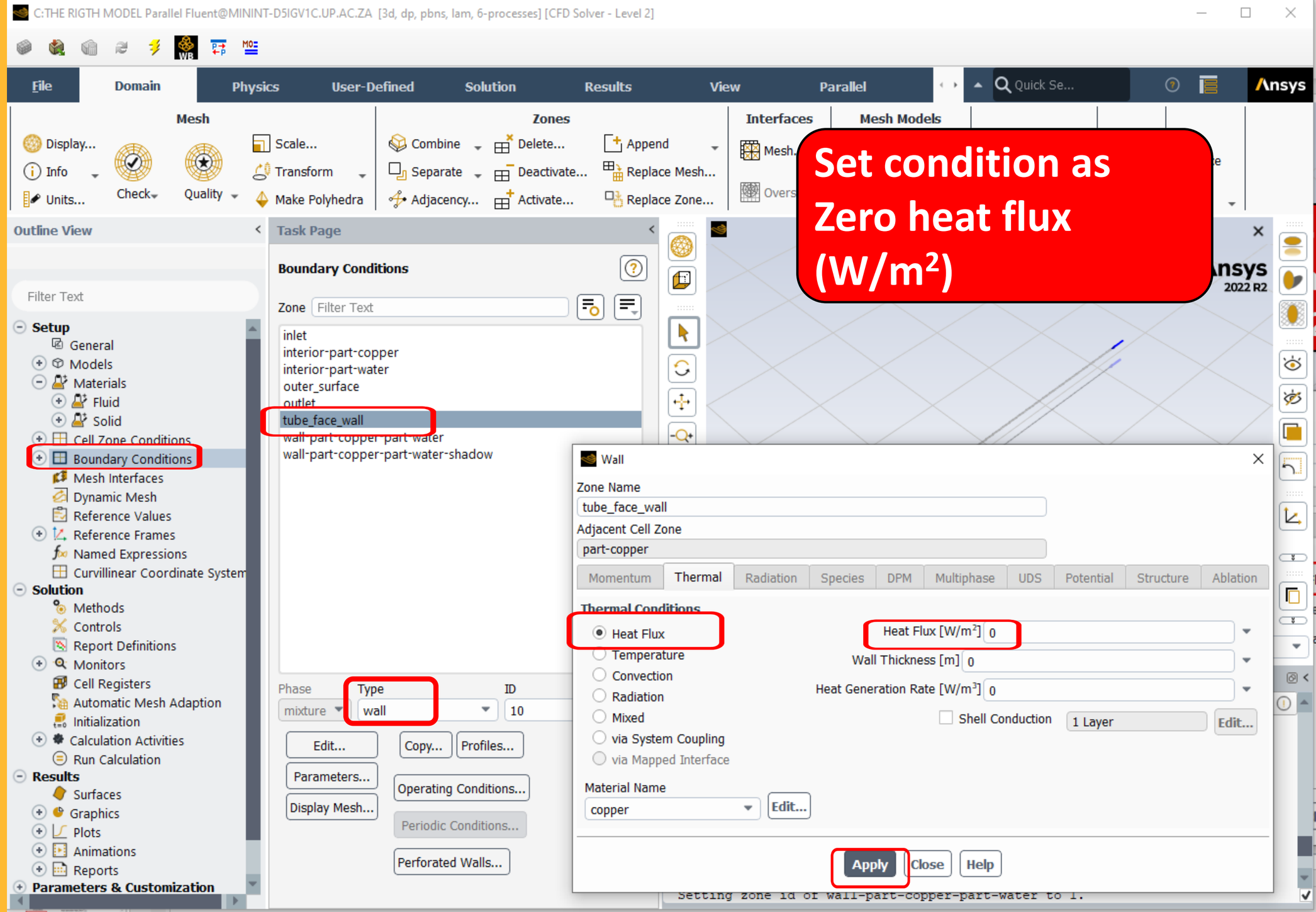
PHYSICS

SETUP



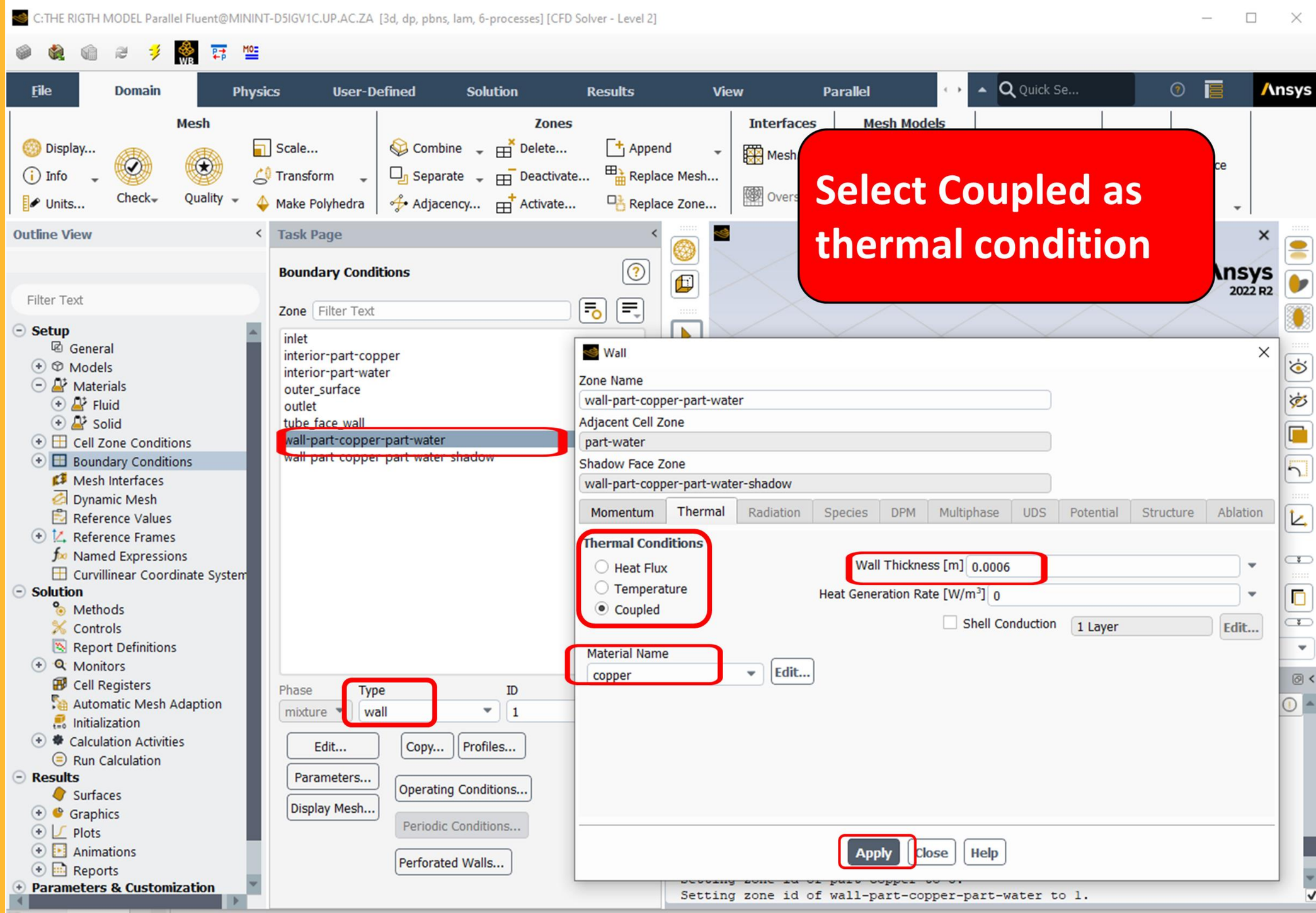
PHYSICS

SETUP



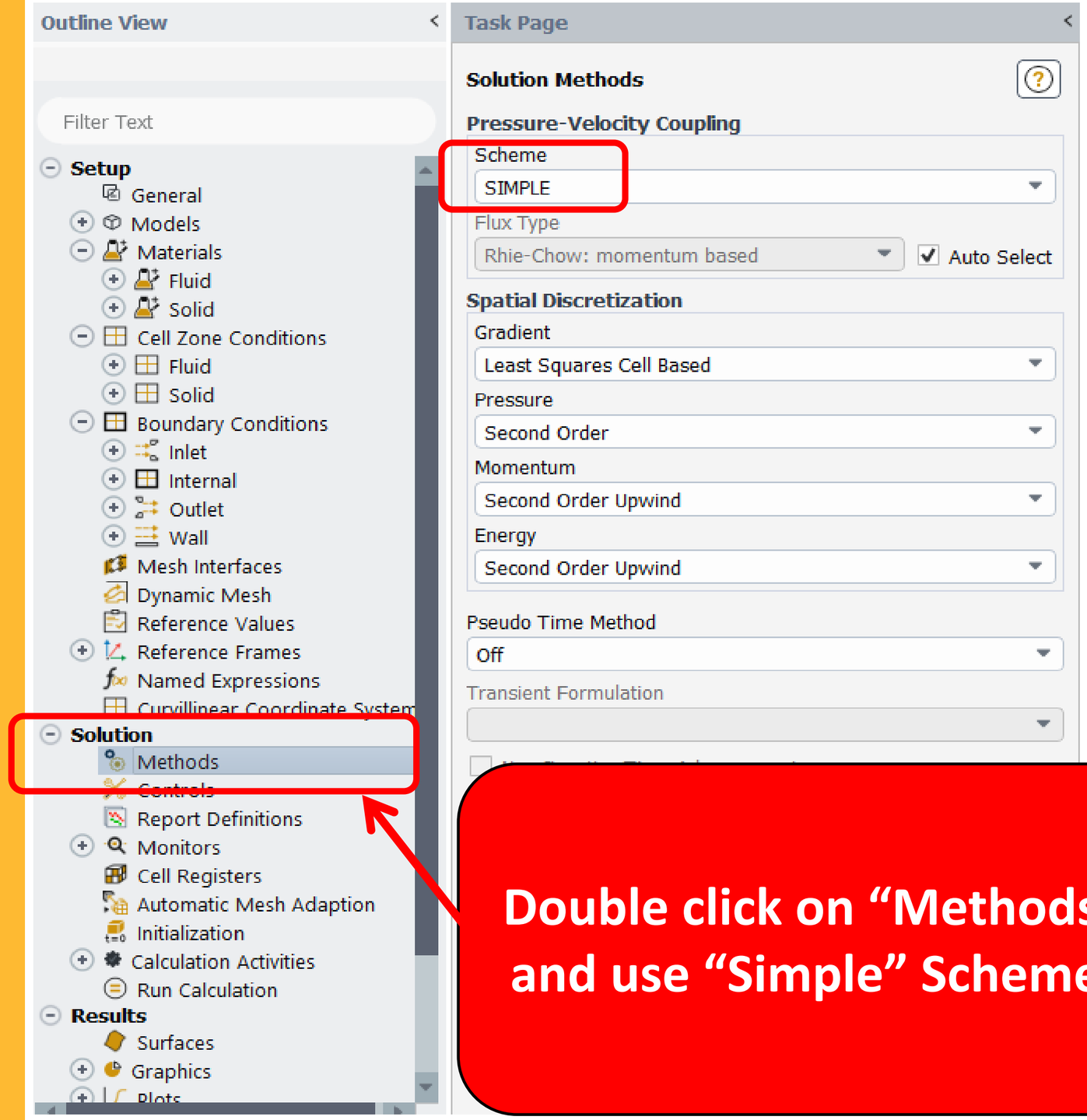
PHYSICS

SETUP



PHYSICS

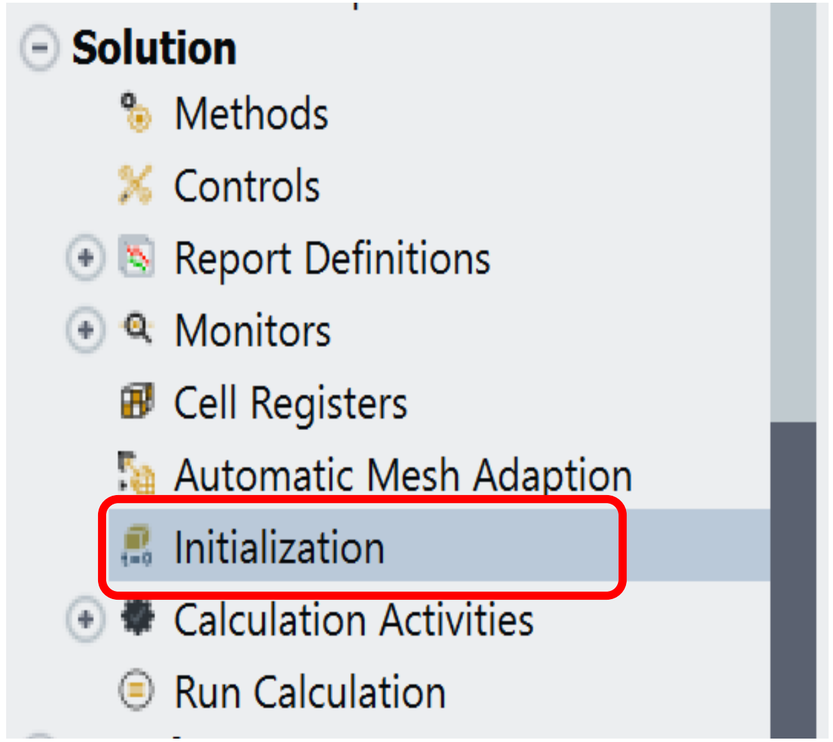
SETUP

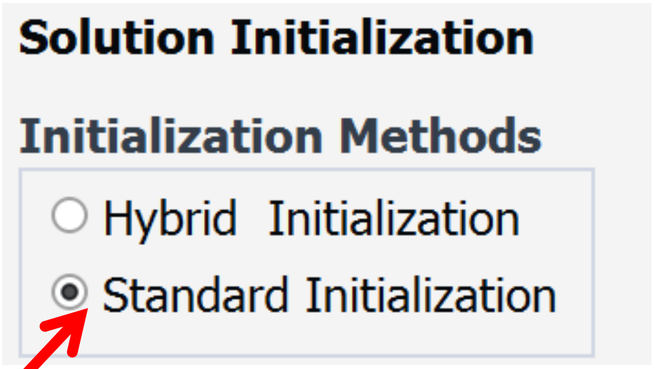


Double click on “Methods” and use “Simple” Scheme.

PHYSICS

SETUP

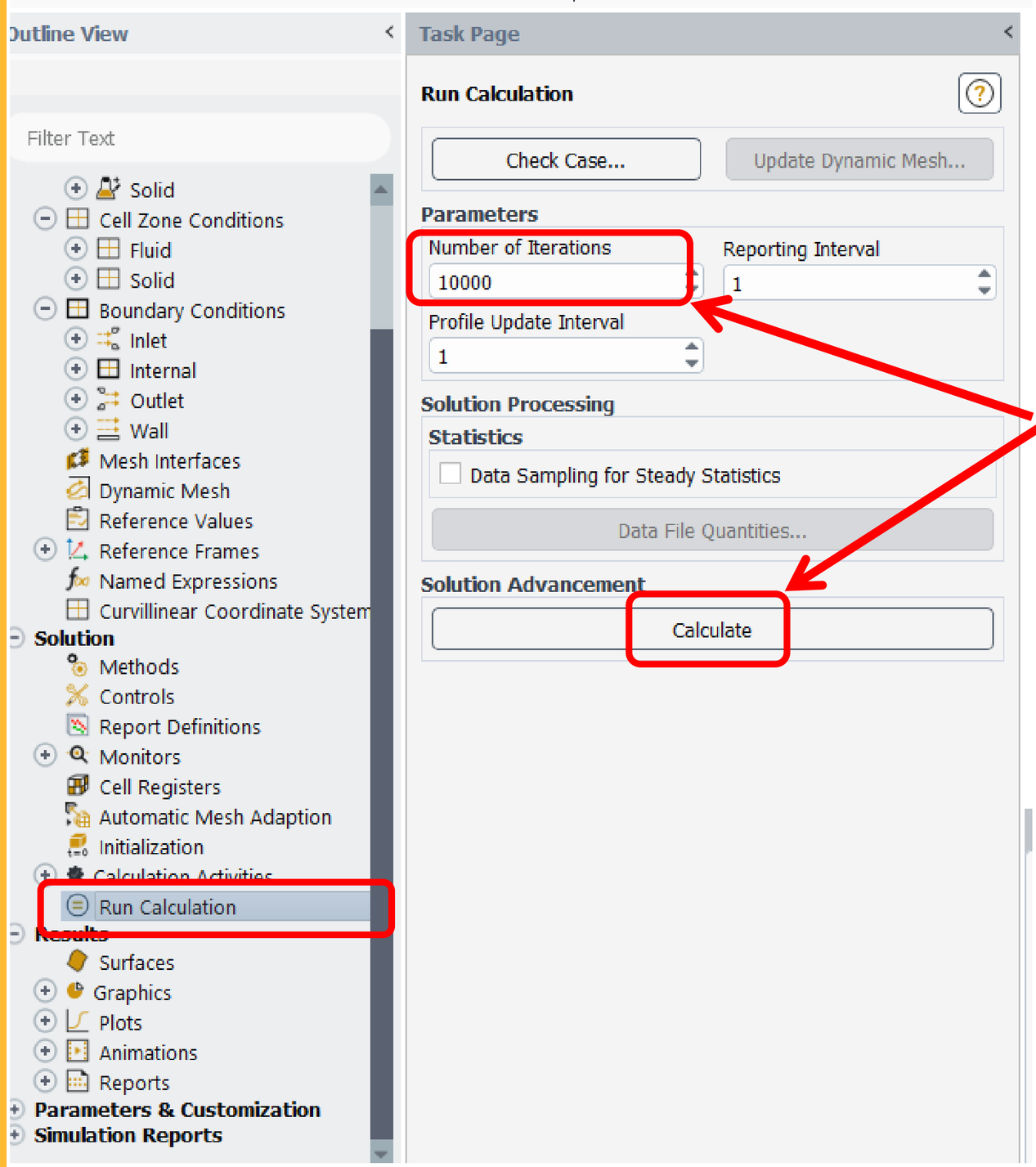




Double click on “Initialization”, select “Standard Initialization” and “Initialize”

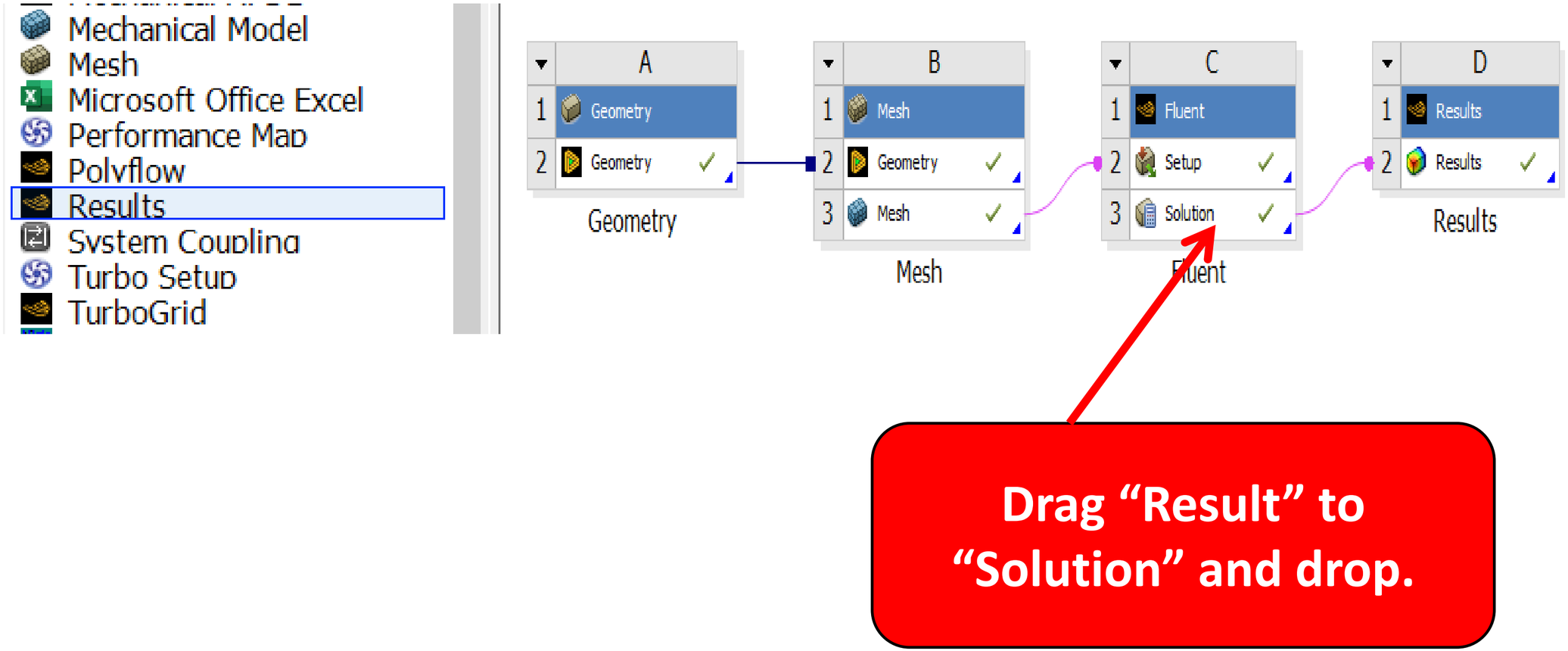
PHYSICS

SETUP



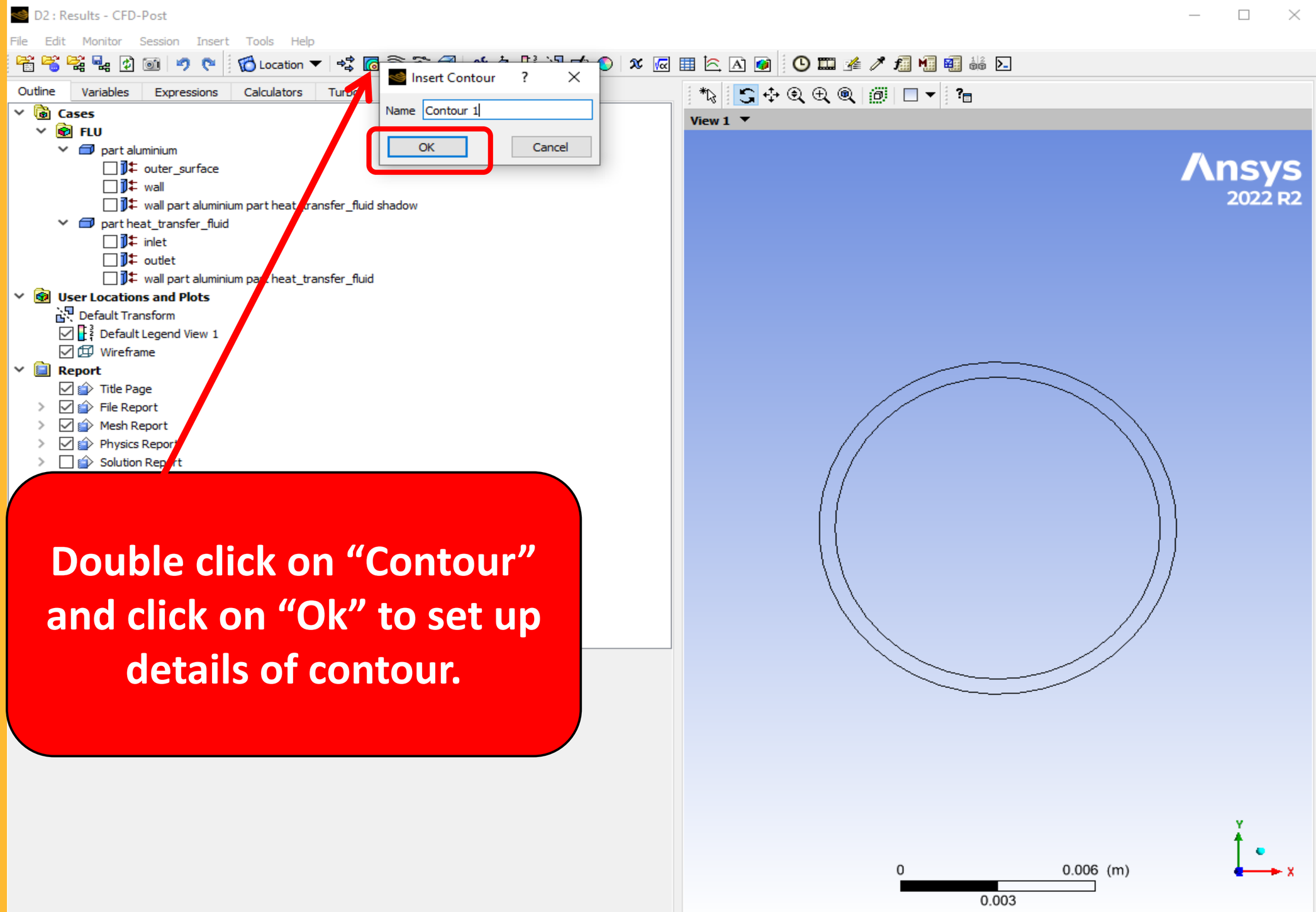
Set Number of Iterations As 10,000 and Click “Calculate”

POST PROCESSING RESULTS


Mechanical Model
Mesh
Microsoft Office Excel
Performance Map
Polyflow
Results
System Coupling
Turbo Setup
TurboGrid
A
1 Geometry
2 Geometry
Geometry
B
1 Mesh
2 Geometry
3 Mesh
Mesh
C
1 Fluent
2 Setup
3 Solution
Fluent
D
1 Results
2 Results
Results
Drag “Result” to “Solution” and drop.

POST

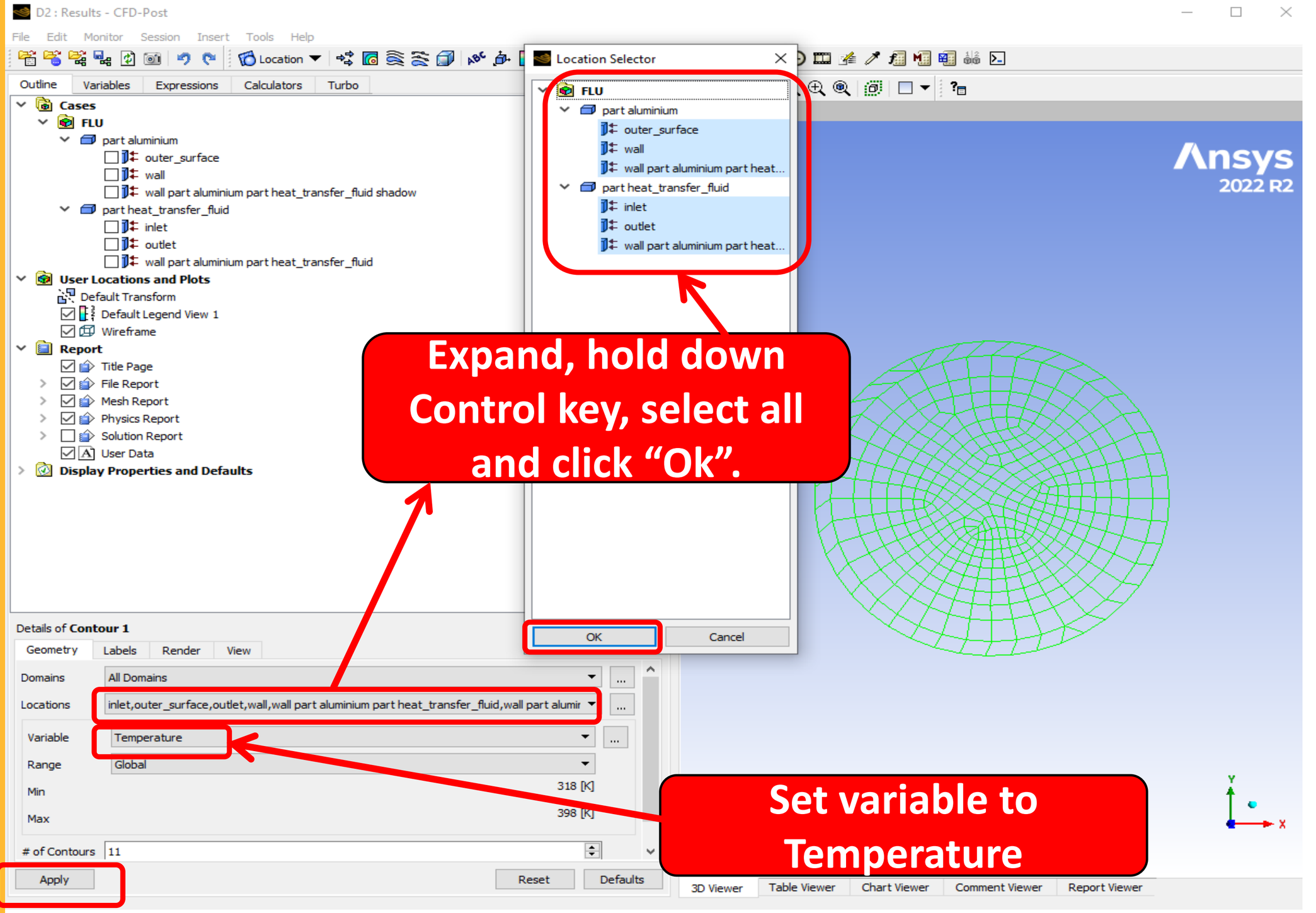
PROCESSING



Double click on “Contour” and click on “Ok” to set up details of contour.

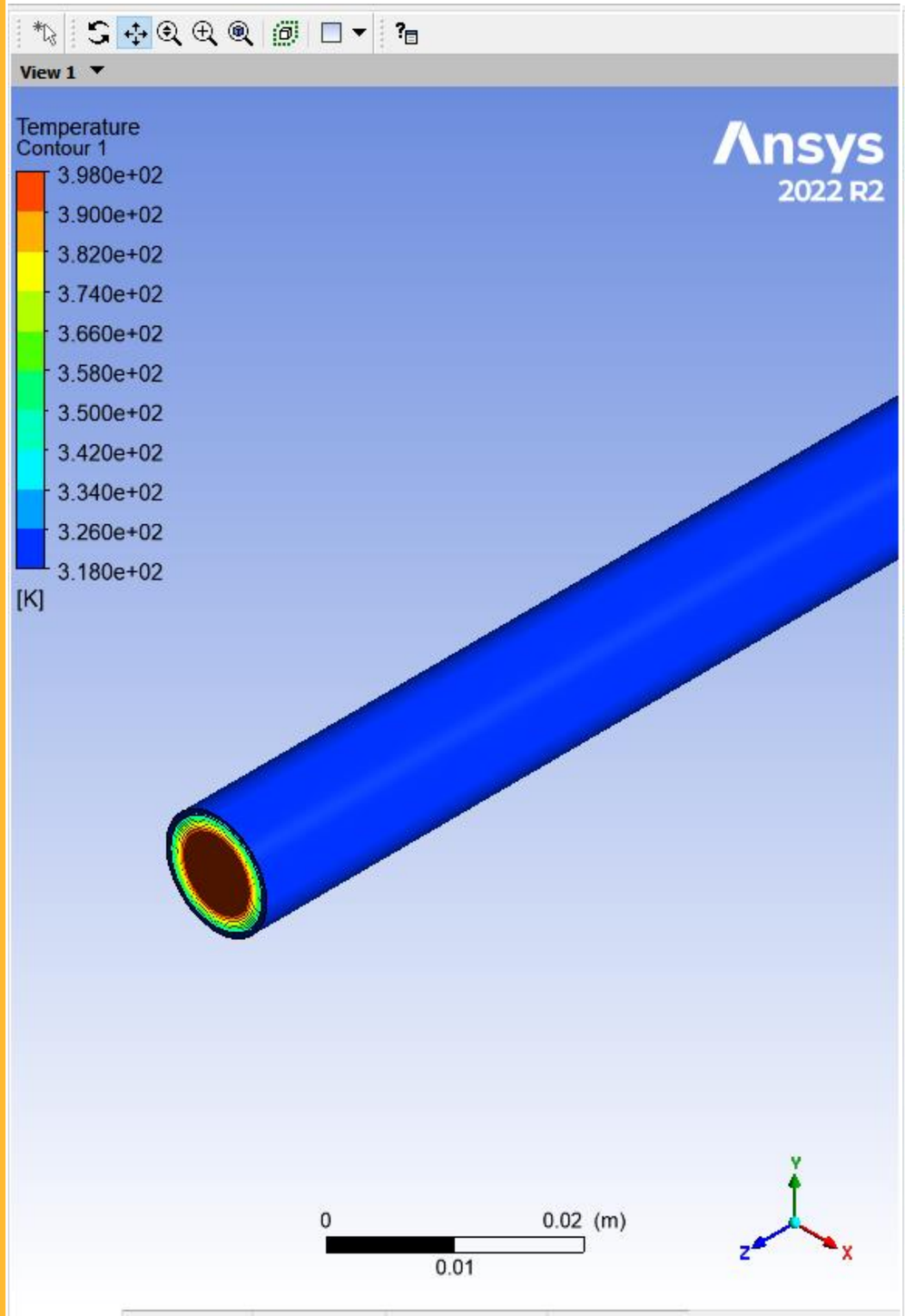
POST

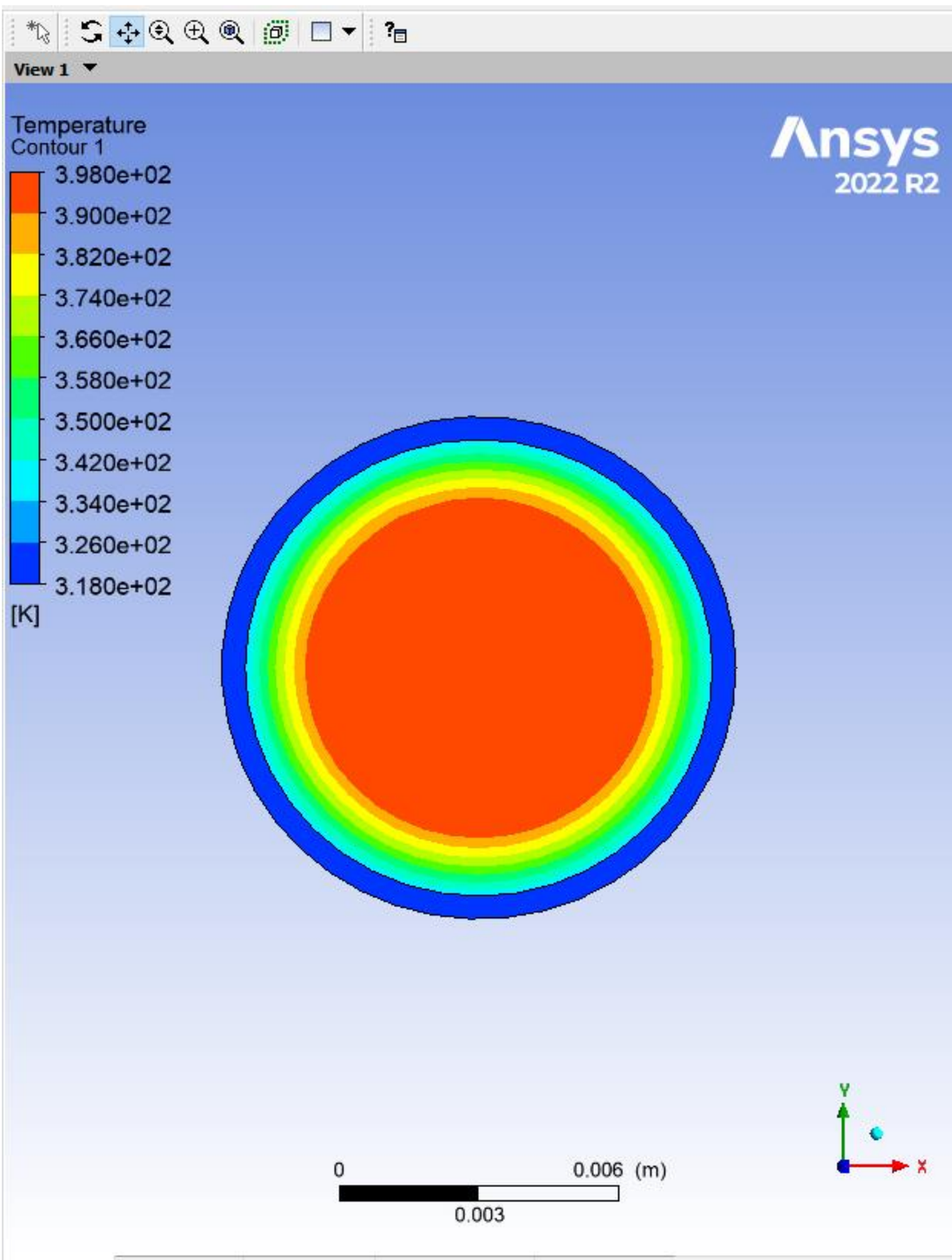
PROCESSING



POST

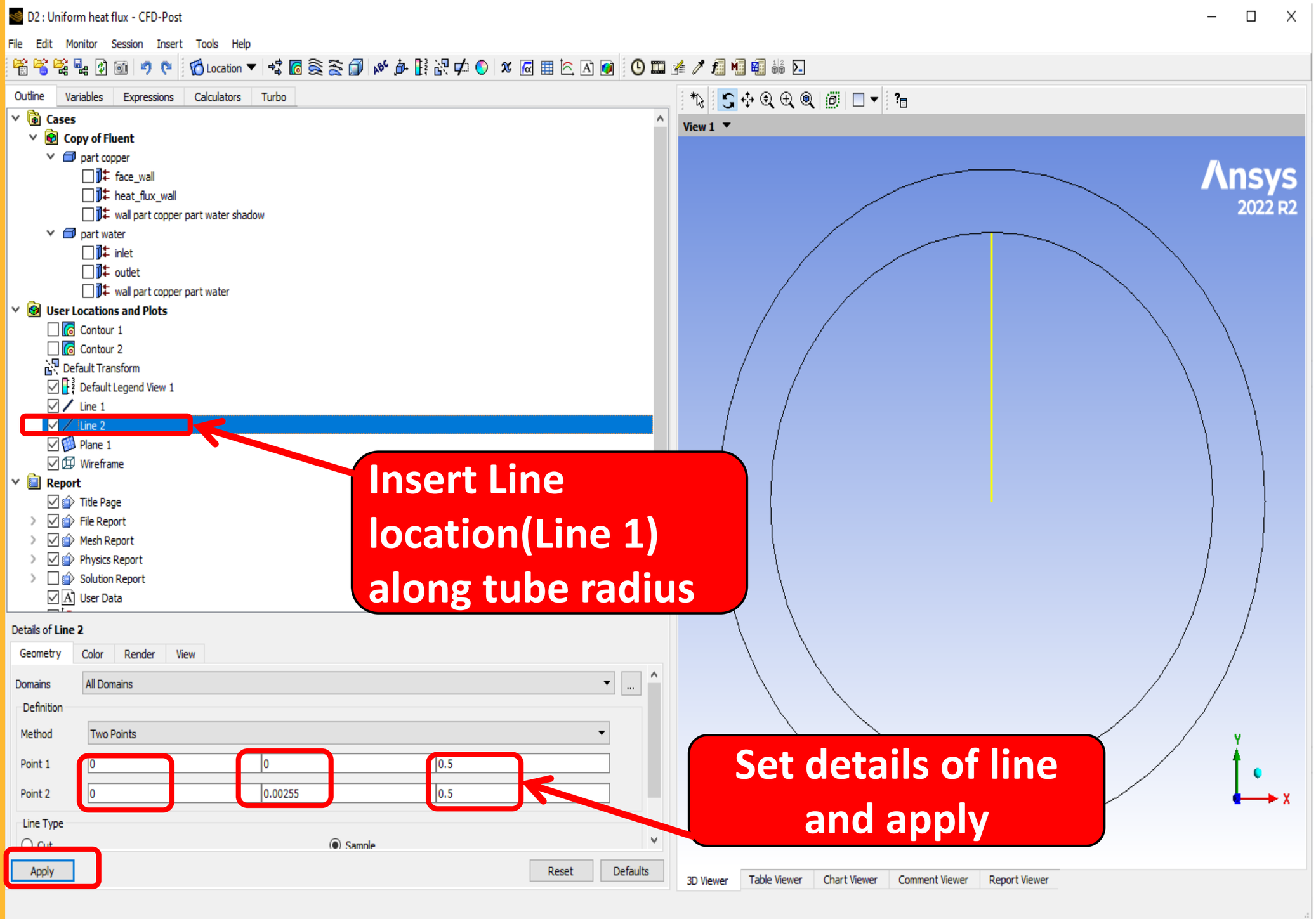
PROCESSING





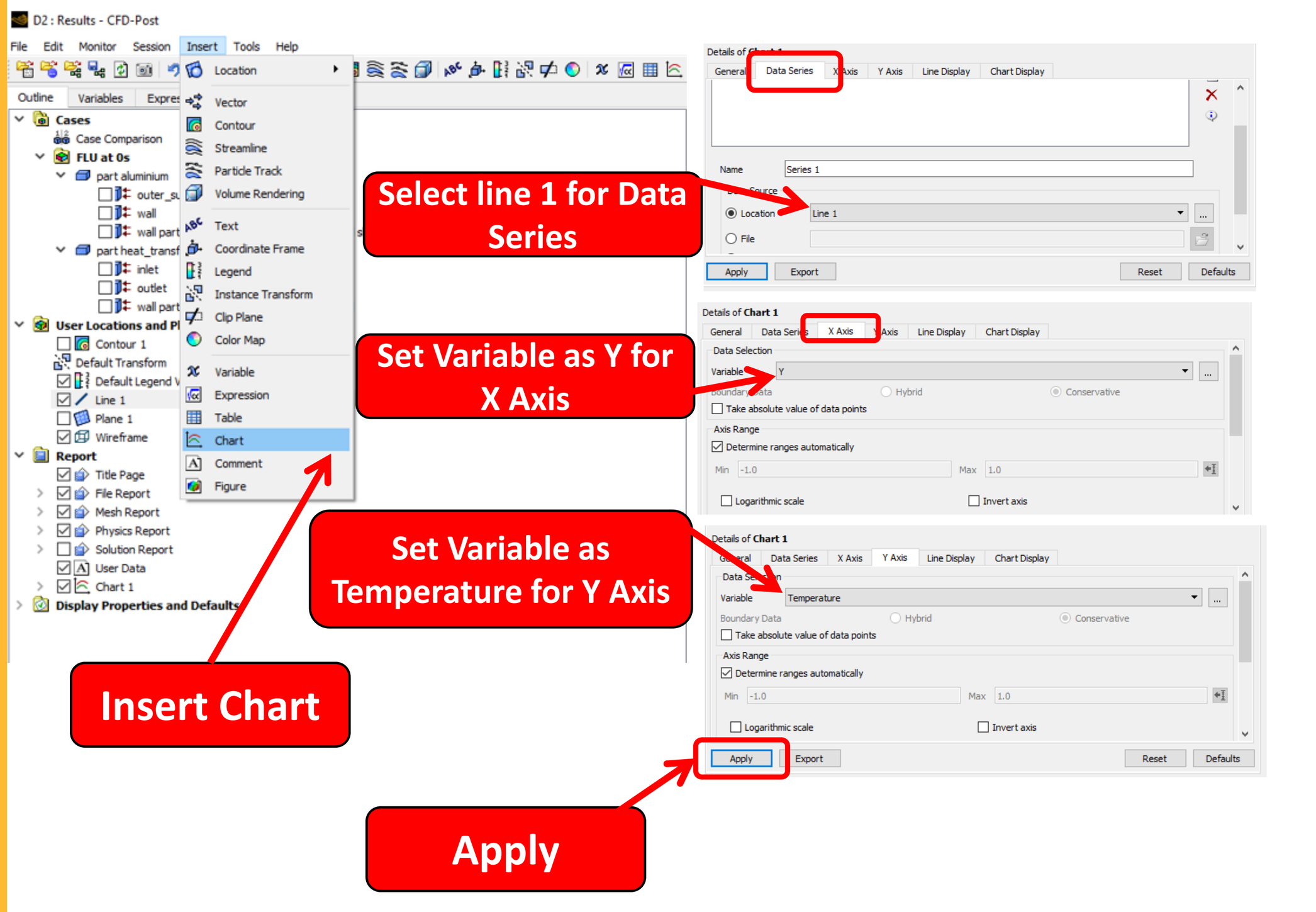
POST

PROCESSING



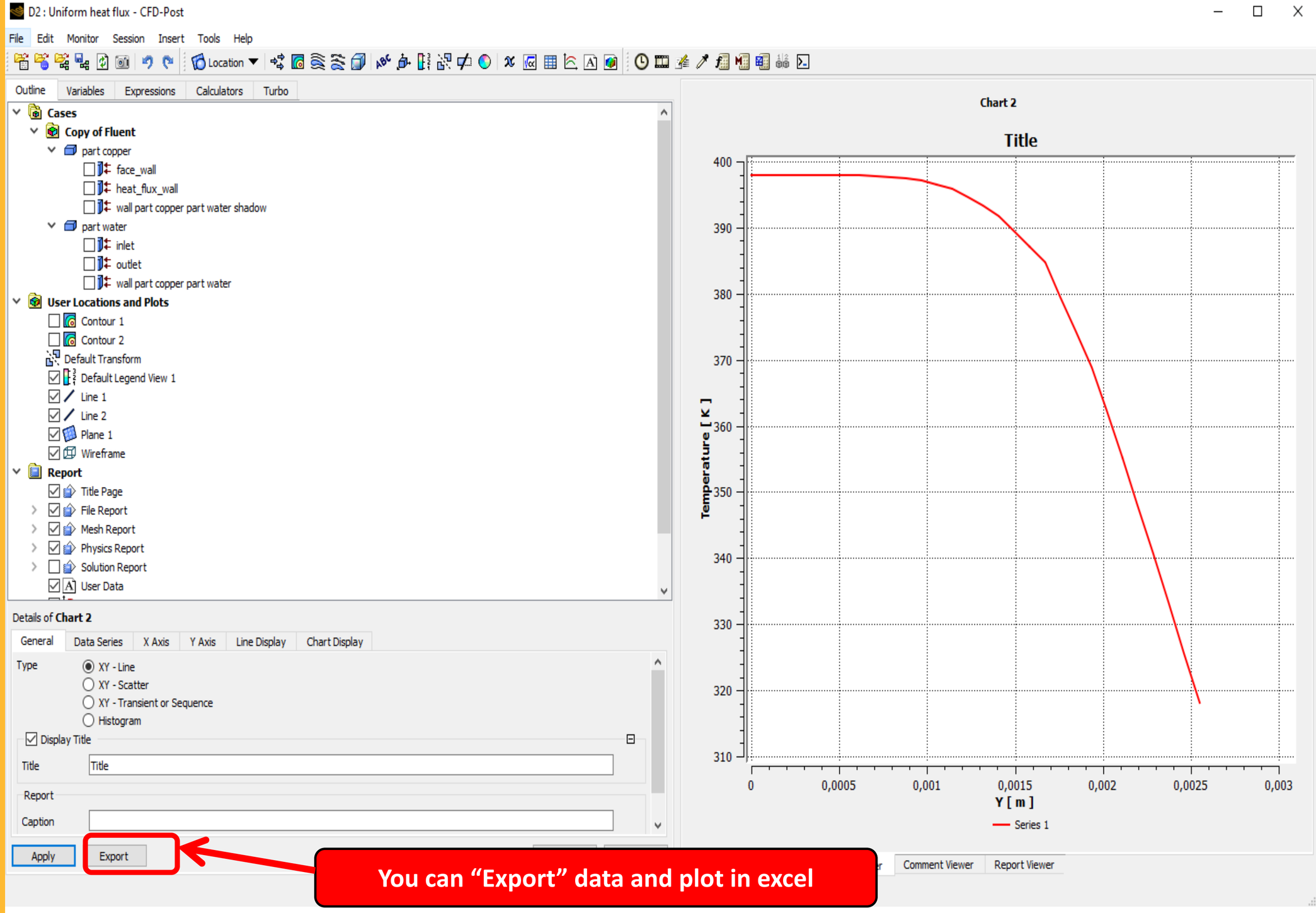
POST

PROCESSING



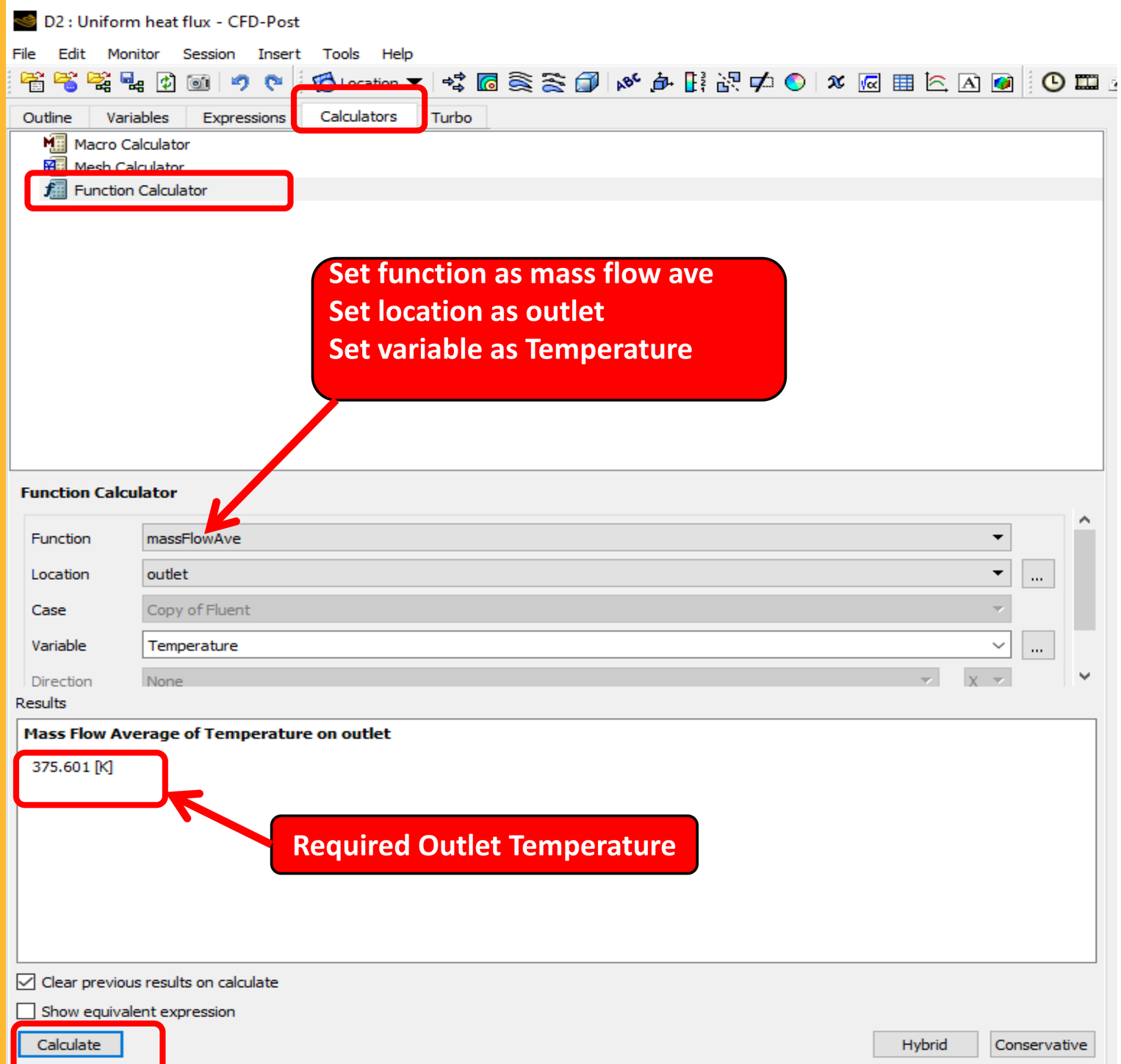
POST

PROCESSING



POST

PROCESSING



SUMMARY

In this third tutorial, you were able to

01 Set up the physics of the problem in the Fluent Solver

02 Analyze the results in the post-processor