

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

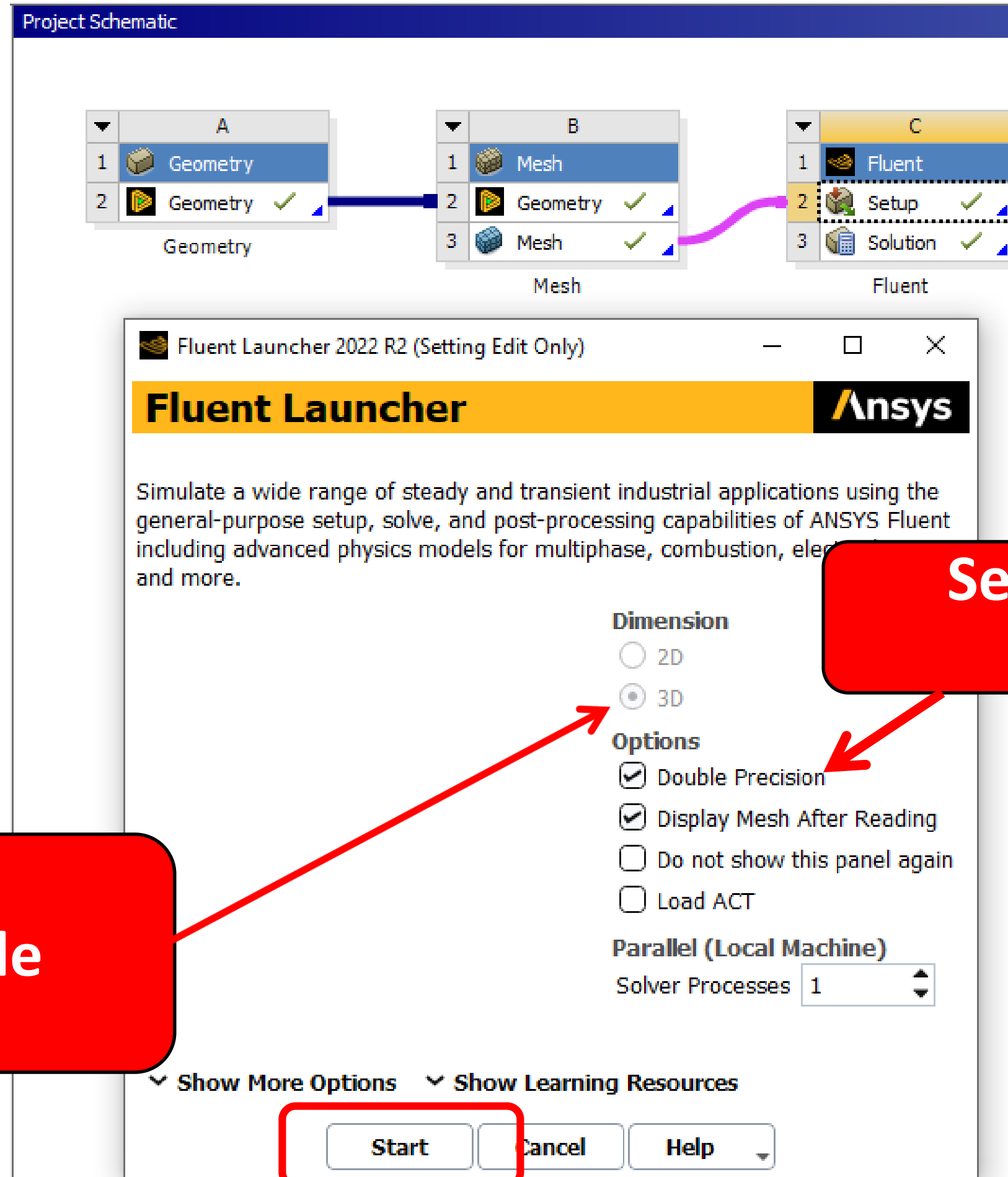
C		
1	Fluent	
2	Setup	✓
3	Solution	✓

Fluent

**Drag “Fluent” to “Mesh”
and drop.**

PHYSICS

SETUP



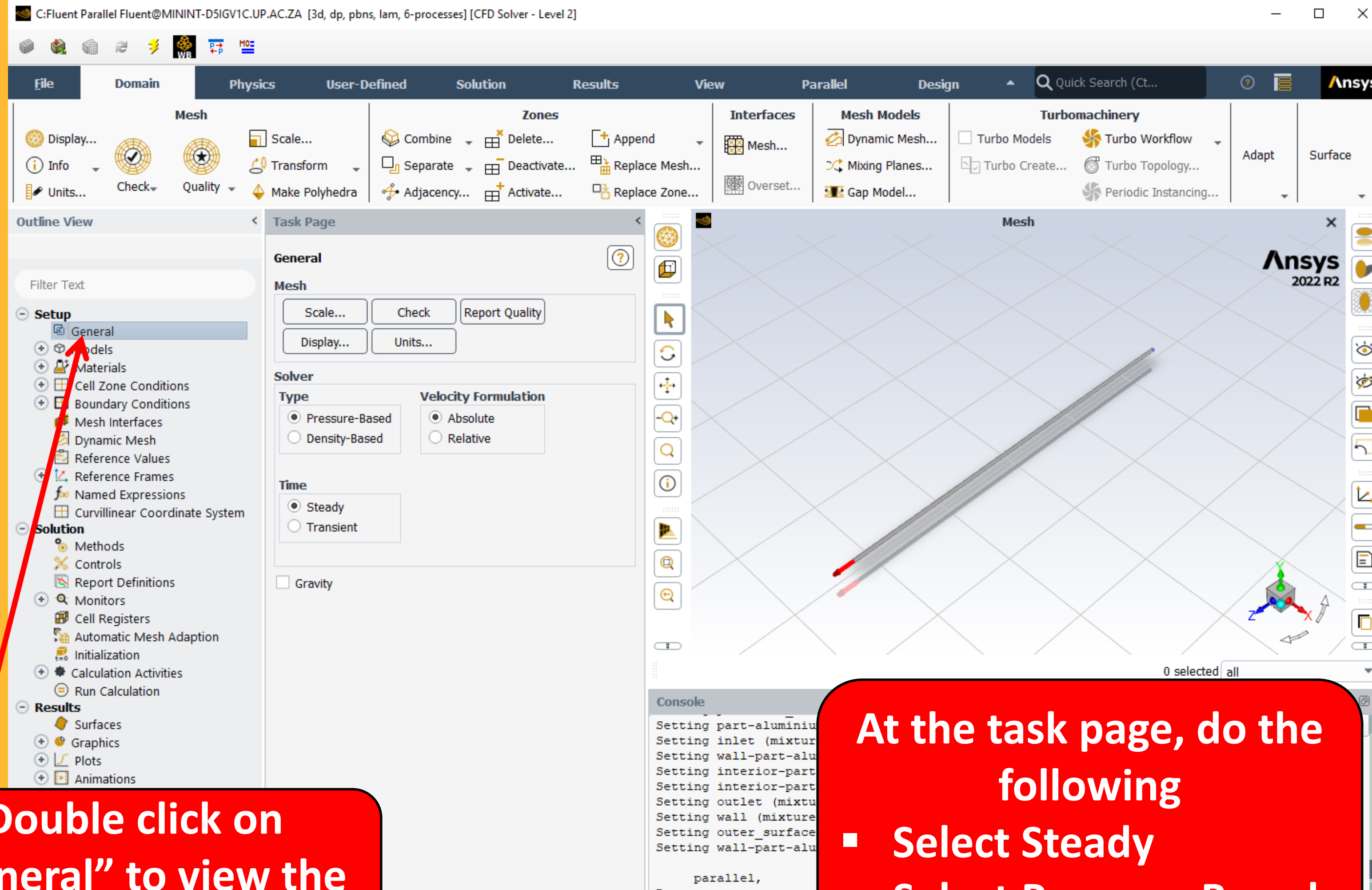
Select 3D mode

**Select Double
precision**

PHYSICS

SETUP

**Double click on
“General” to view the
Task page.**



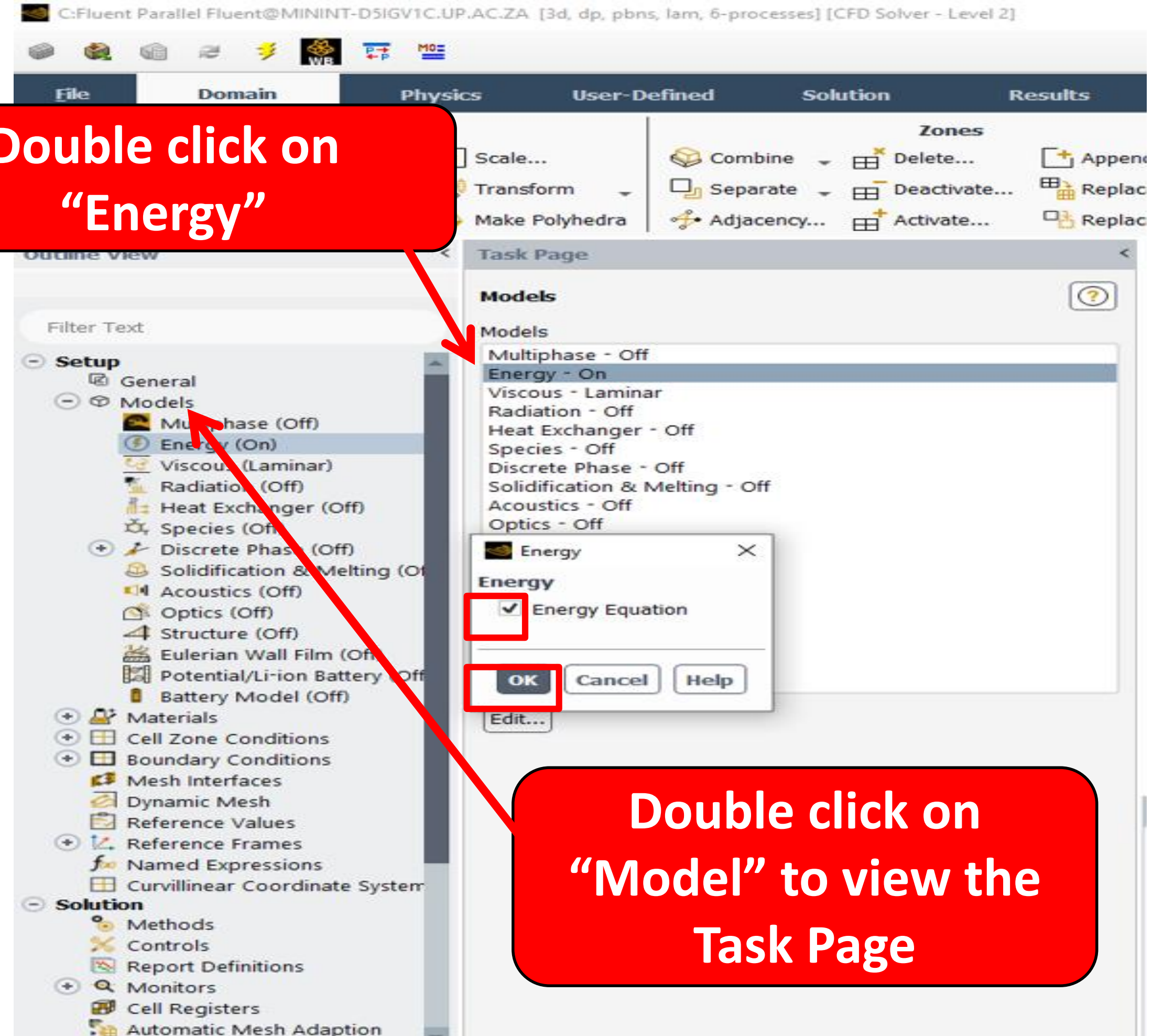
**At the task page, do the
following**

- **Select Steady**
- **Select Pressure Based**

PHYSICS

SETUP

Double click on
“Energy”



Double click on
“Model” to view the
Task Page

PHYSICS

SETUP

Outline View

Filter Text

- Setup
 - General
 - Models
 - Multiphase (Off)
 - Energy (On)
 - Viscous (Laminar)
 - Radiation (Off)
 - Heat Exchanger (Off)
 - Species (Off)
 - Discrete Phase (Off)
 - Solidification & Melting (Off)
 - Acoustics (Off)
 - Optics (Off)
 - Structure (Off)
 - Eulerian Wall Film (Off)
 - Potential/Li-ion Battery (Off)
 - Battery Model (Off)
 - Materials
 - Cell Zone Conditions
 - Boundary Conditions
 - Mesh Interfaces
 - Dynamic Mesh
 - Reference Values
 - Reference Frames
 - Named Expressions
 - Curvilinear Coordinate System
- Solution
 - Methods
 - Controls
 - Report Definitions

Task Page

Models

Models

- Multiphase - Off
- Energy - On
- Viscous - Laminar
- Radiation - Off
- Heat Exchanger - Off
- Species - Off
- Discrete Phase - Off
- Solidification & Melting - Off
- Acoustics - Off
- Optics - Off
- Structure - Off
- Eulerian Wall Film - Off
- Potential/Li-ion Battery - Off
- Battery Model - Off

Edit...

Viscous Model

Model

- ☐ Inviscid
- ☒ Laminar
- ☐ Spalart-Allmaras (1 eqn)
- ☐ k-epsilon (2 eqn)
- ☐ k-omega (2 eqn)
- ☐ Transition k-k1-omega (3 eqn)
- ☐ Transition SST (4 eqn)
- ☐ Reynolds Stress (7 eqn)
- ☐ Scale-Adaptive Simulation (SAS)
- ☐ Detached Eddy Simulation (DES)
- ☐ Large Eddy Simulation (LES)

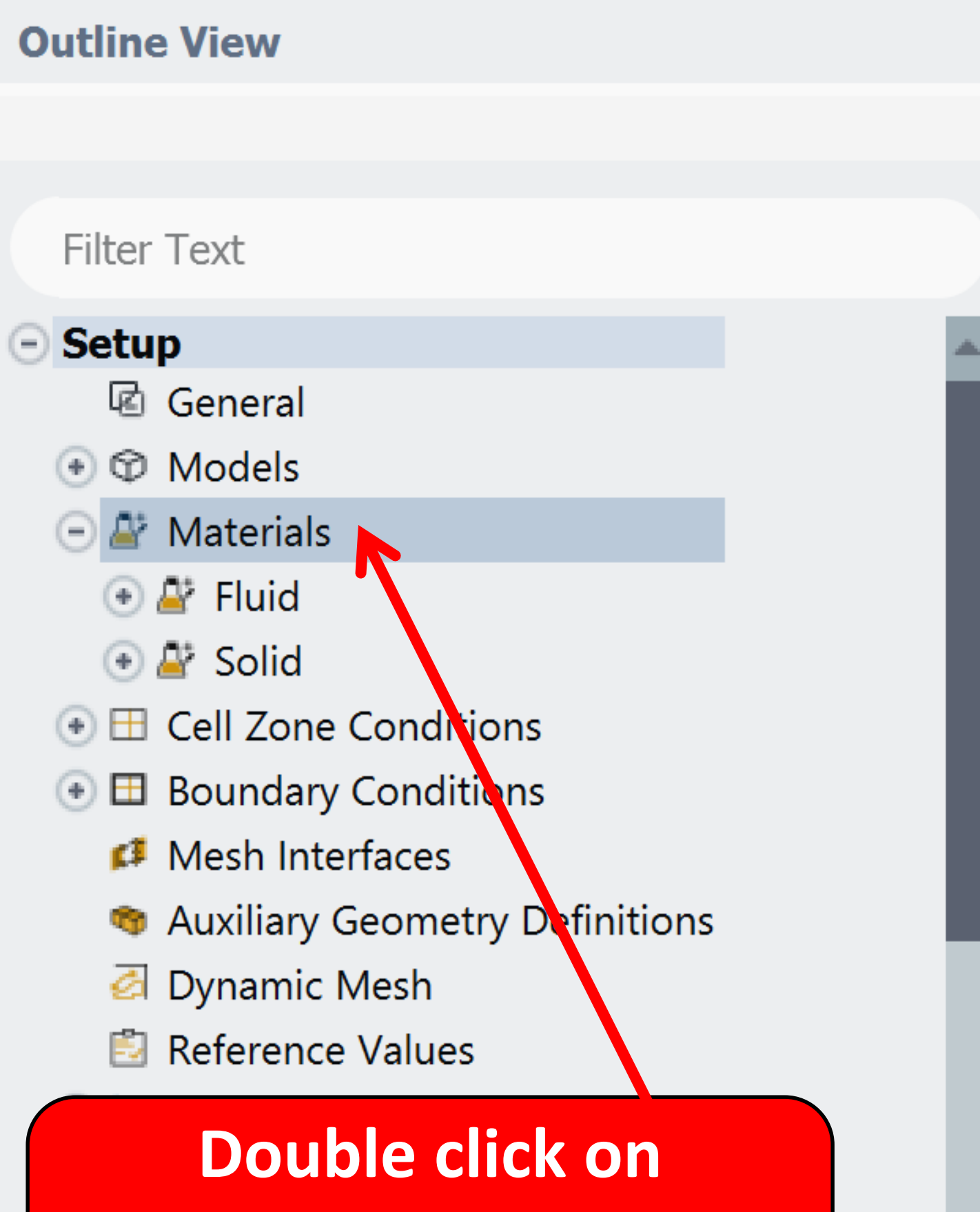
Options

- ☐ Viscous Heating
- ☐ Low-Pressure Boundary Slip

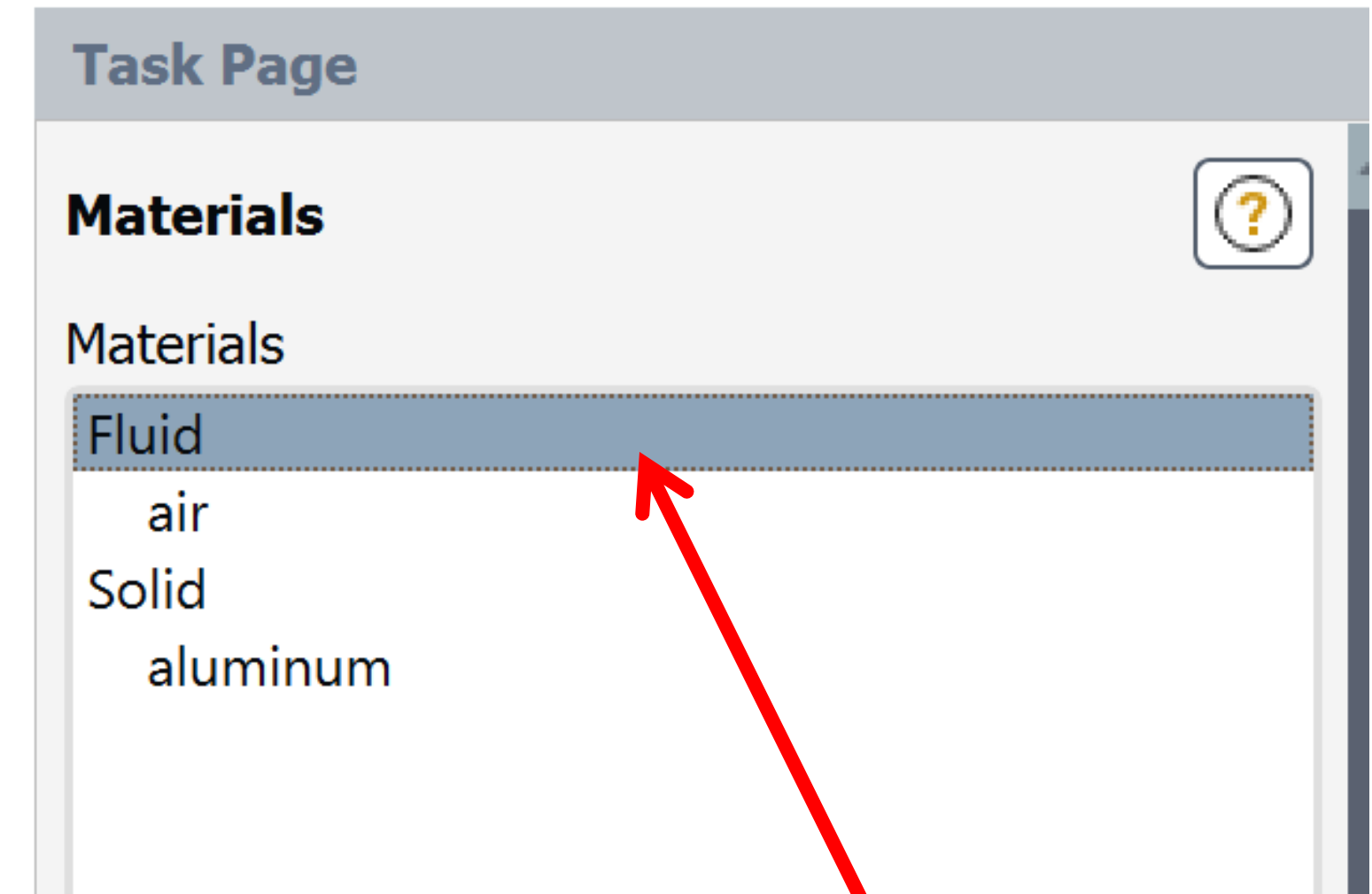
OK Cancel Help

PHYSICS

SETUP



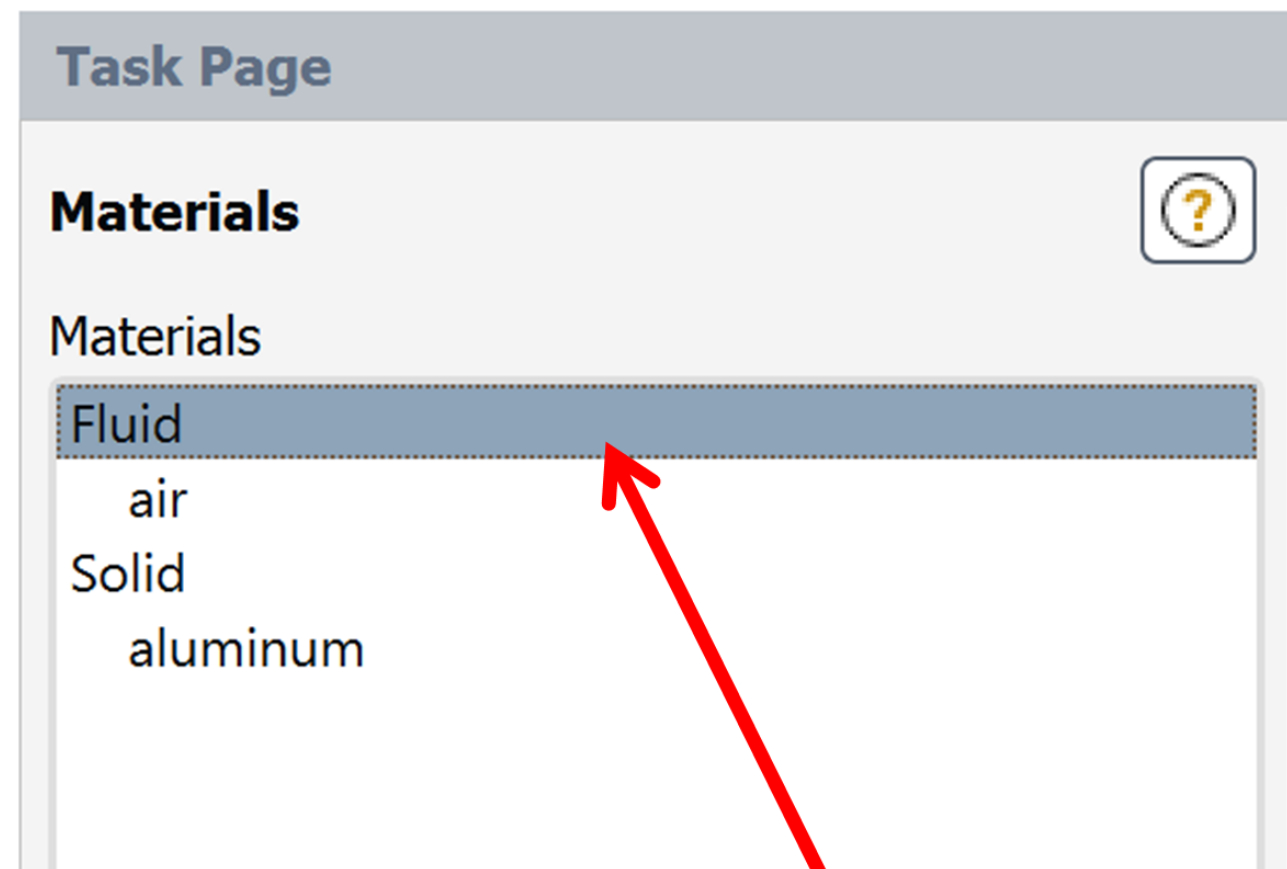
**Double click on
“Materials” to view the
Task page**



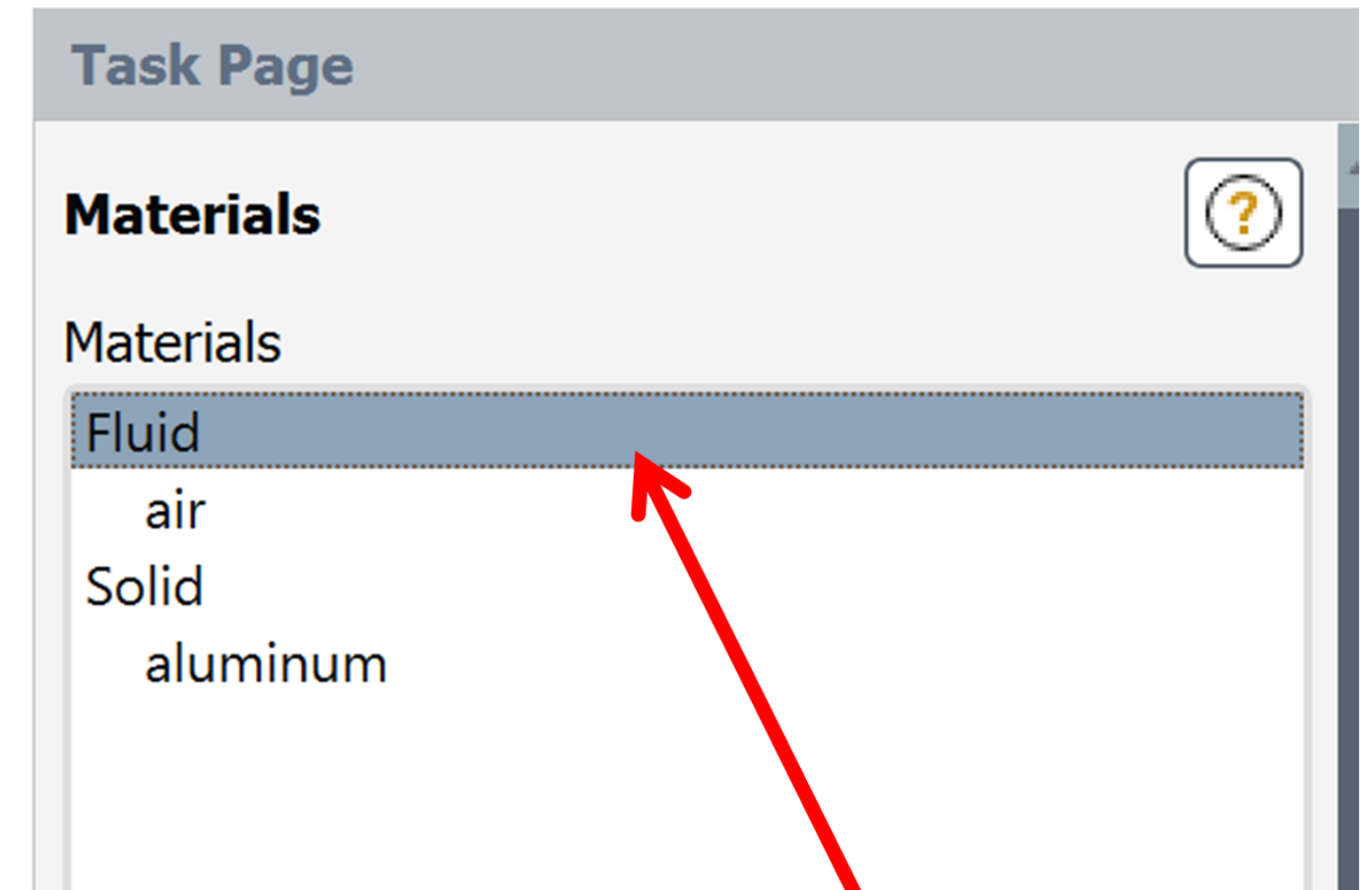
**Double click on “Fluid”
to add the required fluid
from Fluent Database.**

PHYSICS

SETUP



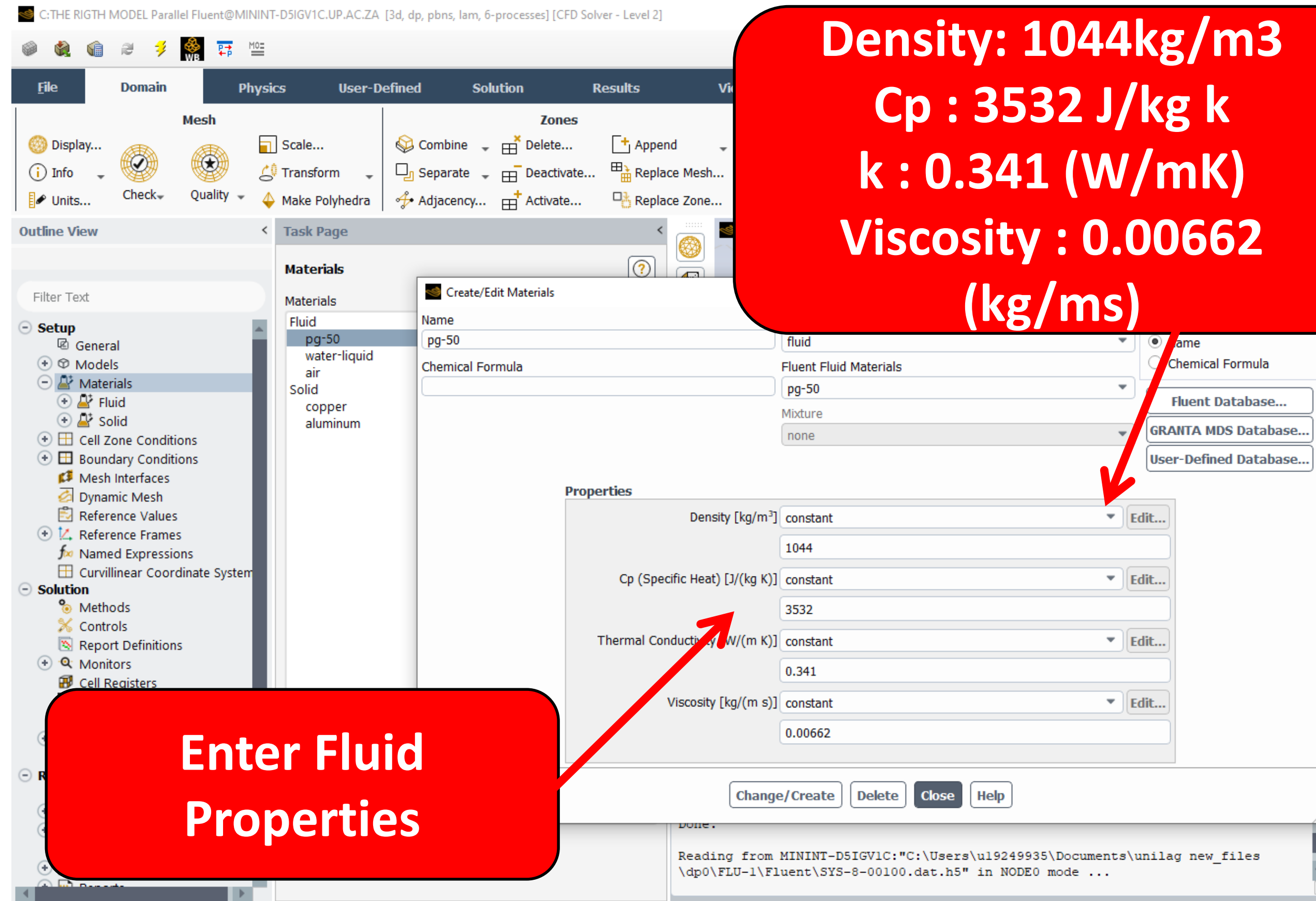
Double click on “Fluid” to add the required fluid properties (Mixture of water and 30% PG).



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

PHYSICS

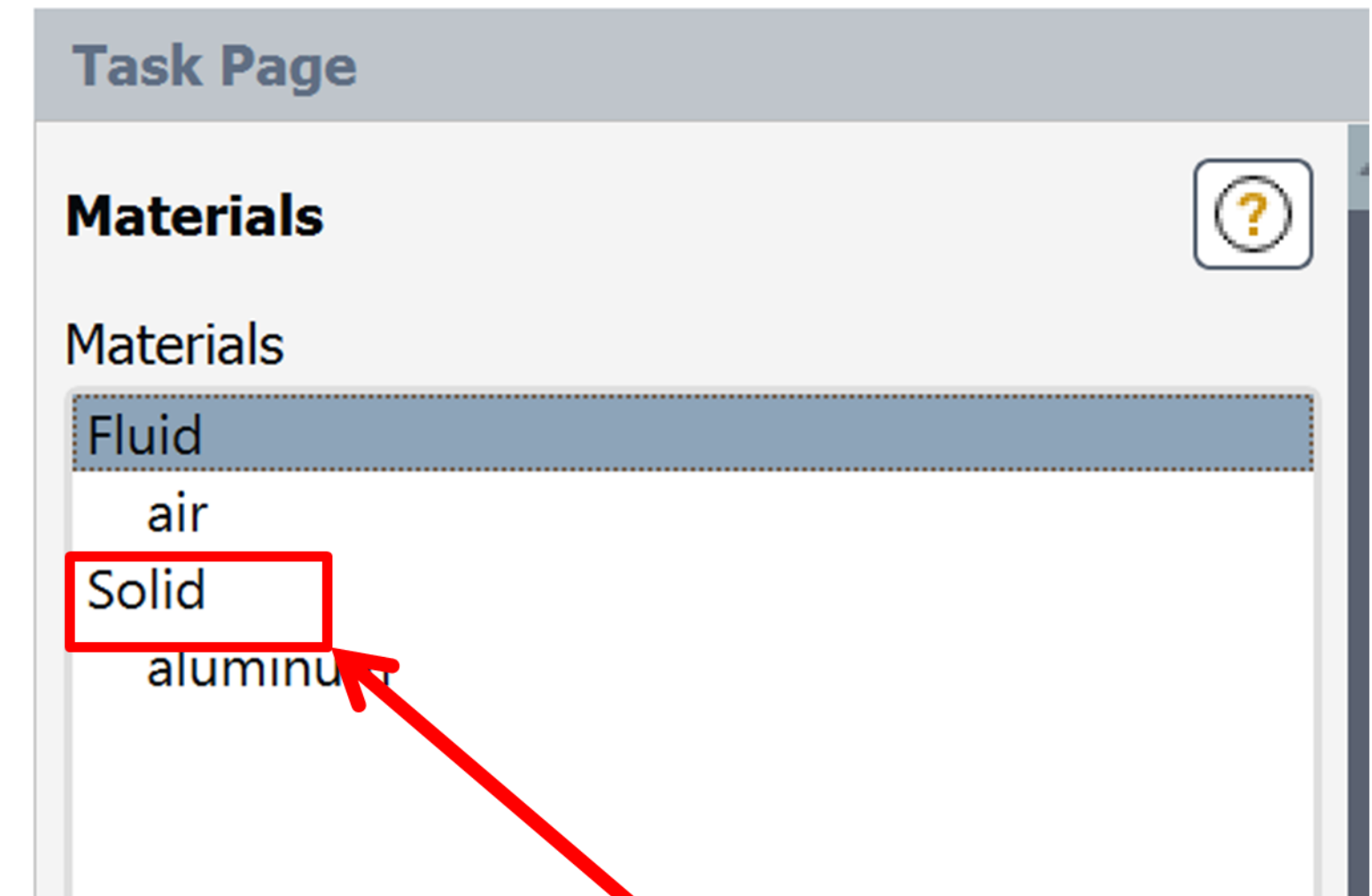
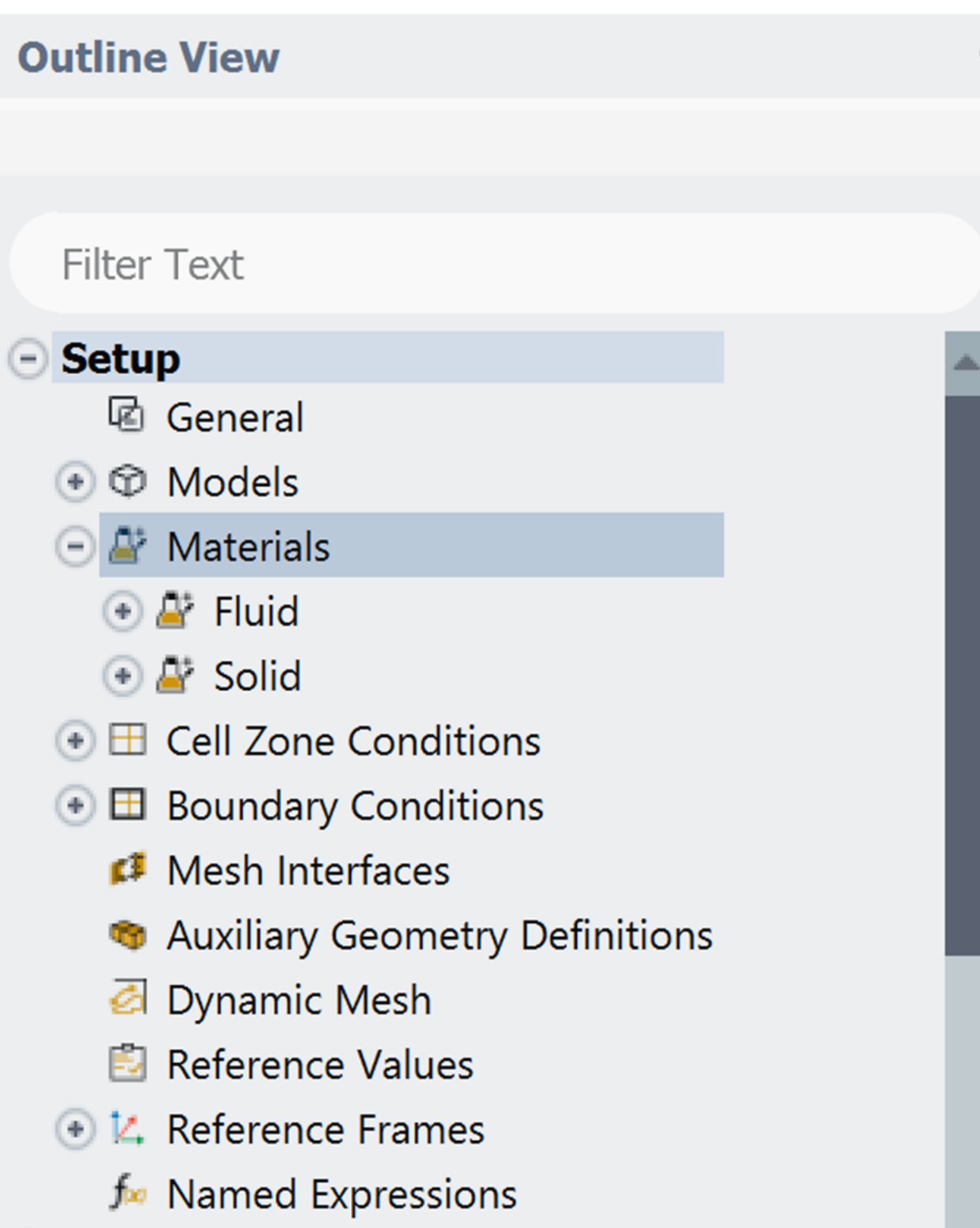
SETUP



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

C:\THE RIGTH MODEL Parallel Fluent@MININT-D5IGV1C.UP.AC.ZA [3d, dp, pbns, lam, 6-processes] [CFD Solver - Level 2]

File Domain **Physics** User-Defined Solution Results View Parallel

Mesh Zones Interfaces Mesh Models

Display... Info Units... Check Quality Scale... Transform Make Polyhedra Combine Separate Adjacency... Delete... Deactivate... Activate... Append Replace Mesh... Replace Zone... Mesh... Overset... Dynamic Mesh... Mixing Planes... Gap Model... Turbomachinery Adapt Surface

Outline View Task Page

Materials

Filter Text

Setup

- General
- Models
- Materials
- Fluid
- Solid
- Cell Zone Conditions
- Boundary Conditions
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- Reference Frames
- Named Expressions
- Curvilinear Coordinate System

Solution

- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Automatic Mesh Adaption
- Initialization
- Calculation Activities

Create/Edit Materials

Name: copper

Material Type: solid

Chemical Formula: cu

Fluent Solid Materials: copper (cu)

Mixture: none

Order Materials by: Name

Properties

Density [kg/m³]: constant 8978

Cp (Specific Heat) [J/(kg K)]: constant 381

Thermal Conductivity [W/(m K)]: constant 401

Change/Create Delete Close Help

Enter Solid Properties

Reading from MININT-D5IGV1C:"C:\Users\ul9249935\Documents\unilag new_files\dp0\FLU-1\Fluent\SYS-8-00100.dat.h5" in NODE0 mode ...

PHYSICS

SETUP

C:\THE RIGTH MODEL Parallel Fluent@MININT-D5IGV1C.UP.AC.ZA [3d, dp, pbns, lam, 6-processes] [CFD Solver - Level 2]

File Domain Physics User-Defined Solution Results View Parallel

Mesh Zones Interfaces Mesh Models

Display... Info Units... Check Quality Scale... Transform Make Polyhedra Combine Separate Adjacency... Delete... Deactivate... Activate... Append Replace Mesh... Replace Zone... Mesh... Overset... Dynamic Mesh... Mixing Planes... Gap Model... Turbomachinery Adapt Surface

Outline View Task Page

Filter Text

Setup

- General
- Models
- Materials
 - Fluid
 - Solid
- Cell Zone Conditions
 - Boundary Conditions**
 - Inlet
 - Internal
 - Outlet
 - Wall
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- Reference Frames
- Named Expressions
- Curvilinear Coordinate System

Solution

- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Automatic Mesh Adaption
- Initialization
- Calculation Activities
- Run Calculation

Results

- Surfaces
- Graphics

Boundary Conditions

Zone Filter Text

- face_wall
- heat_flux_wall
- inlet**
- interior-part-copper
- interior-part-water
- outlet
- wall-part-copper-part-water
- wall-part-copper-part-water-shadow

Phase Type ID

Phase	Type	ID
mixture	velocity-inlet	8

Edit... Copy... Profiles... Parameters... Operating Conditions... Display Mesh... Periodic Conditions... Perforated Walls...

Velocity Inlet

Zone Name inlet

Momentum Thermal Radiation Species DPM Multiphase Potential Structure UDS

Velocity Specification Method Magnitude, Normal to Boundary

Reference Frame Absolute

Velocity Magnitude [m/s] 0.3114

Supersonic/Initial Gauge Pressure [Pa] 0

Apply Close Help

Done.

Preparing mesh for display...

Done.

Reading from MININT-D5IGV1C:"C:\Users\ul9249935\Documents\unilag new_files\dp0\FLU-1\Fluent\SYS-8-00100.dat.h5" in NODE0 mode ...

Enter Velocity inlet value 0.3114m/s

PHYSICS

SETUP

C:\THE RIGH MODEL Parallel Fluent@MININT-D5IGV1C.UP.AC.ZA [3d, dp, pbns, lam, 6-processes] [CFD Solver - Level 2]

File Domain Physics User-Defined Solution Results View Parallel

Mesh Zones Interfaces Mesh Models

Display... Info Units... Check Quality Scale... Transform Make Polyhedra Combine Separate Adjacency... Delete... Deactivate... Activate... Append Replace Mesh... Replace Zone... Mesh... Overset... Dynamic Mesh... Mixing Planes... Gap Model... Turbomachinery Adapt Surface

Outline View Task Page

Filter Text

Setup

- General
- Models
- Materials
 - Fluid
 - Solid
- Cell Zone Conditions
- Boundary Conditions**
 - Inlet
 - Internal
 - Outlet
 - Wall
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- Reference Frames
- Named Expressions
- Curvilinear Coordinate System

Solution

- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Automatic Mesh Adaption
- Initialization
- Calculation Activities
- Run Calculation

Results

- Surfaces
- Graphics

Boundary Conditions

Zone Filter Text

- face_wall
- heat_flux_wall
- inlet**
- interior-part-copper
- interior-part-water
- outlet
- wall-part-copper-part-water
- wall-part-copper-part-water-shadow

Phase Type ID

mixture velocity-inlet 8

Edit... Copy... Profiles... Parameters... Operating Conditions... Display Mesh... Periodic Conditions... Perforated Walls...

Velocity Inlet

Zone Name inlet

Momentum **Thermal** Radiation Species DPM Multiphase Potential Structure UDS

Temperature [K] 398

Apply Close Help

Done.

Preparing mesh for display...

Done.

Reading from MININT-D5IGV1C:"C:\Users\ul9249935\Documents\unilag new_files\dp0\FLU-1\Fluent\SYS-8-00100.dat.h5" in NODE0 mode ...

Enter Inlet temperature value 398K

PHYSICS

SETUP

C:\THE RIGTH MODEL Parallel Fluent@MININT-D5IGV1C.UP.AC.ZA [3d, dp, pbns, lam, 6-processes] [CFD Solver - Level 2]

File Domain Physics User-Defined Solution Results View Parallel

Mesh Zones Interfaces Mesh Models

Display... Info Units... Check Quality Scale... Transform Make Polyhedra Combine Separate Adjacency... Delete... Deactivate... Activate... Append Replace Mesh... Replace Zone...

Outline View Task Page

Filter Text

Setup

- General
- Models
- Materials
- Fluid
- Solid
- Cell Zone Conditions
- Boundary Conditions**
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- Reference Frames
- Named Expressions
- Curvilinear Coordinate System

Solution

- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Automatic Mesh Adaption
- Initialization
- Calculation Activities
- Run Calculation

Results

- Surfaces
- Graphics
- Plots
- Animations
- Reports

Parameters & Customization

Boundary Conditions

Zone Filter Text

- inlet
- interior-part-copper
- interior-part-water
- outer_surface**
- outlet
- tube_face_wall
- wall-part-copper-part-water
- wall-part-copper-part-water-shadow

Phase mixture Type wall ID 11

Edit... Copy... Profiles... Parameters... Display Mesh... Operating Conditions... Periodic Conditions... Perforated Walls...

Wall

Zone Name outer_surface

Adjacent Cell Zone part-copper

Momentum Thermal Radiation Species DPM Multiphase UDS Potential Structure Ablation

Thermal Conditions

- Heat Flux
- Temperature**
- Convection
- Radiation
- Mixed
- via System Coupling
- via Mapped Interface

Temperature [K] 318

Wall Thickness [m] 0

Heat Generation Rate [W/m³] 0

Shell Conduction 1 Layer Edit...

Material Name copper Edit...

Apply Close Help

Setting zone id of part-copper to 5.
Setting zone id of wall-part-copper-part-water to 1.

PHYSICS

SETUP

C:\THE RIGTH MODEL Parallel Fluent@MININT-D5IGV1C.UP.AC.ZA [3d, dp, pbns, lam, 6-processes] [CFD Solver - Level 2]

File Domain Physics User-Defined Solution Results View Parallel

Mesh Zones Interfaces Mesh Models

Display... Info Units... Check Quality Scale... Transform Make Polyhedra Combine Separate Adjacency... Delete... Deactivate... Activate... Append Replace Mesh... Replace Zone...

Outline View Task Page

Filter Text

Setup

- General
- Models
- Materials
- Fluid
- Solid
- Cell Zone Conditions
- Boundary Conditions**
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- Reference Frames
- Named Expressions
- Curvilinear Coordinate System

Solution

- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Automatic Mesh Adaption
- Initialization
- Calculation Activities
- Run Calculation

Results

- Surfaces
- Graphics
- Plots
- Animations
- Reports

Parameters & Customization

Boundary Conditions

Zone Filter Text

- inlet
- interior-part-copper
- interior-part-water
- outer surface
- outlet**
- tube_face_wall
- wall-part-copper-part-water
- wall-part-copper-part-water-shadow

Phase mixture

Type **pressure-outlet** ID 9

Edit... Copy... Profiles... Parameters... Display Mesh... Operating Conditions... Periodic Conditions... Perforated Walls...

Pressure Outlet

Zone Name **outlet**

Momentum Thermal Radiation Species DPM Multiphase Potential Structure UDS

Backflow Reference Frame Absolute

Gauge Pressure [Pa] **0**

Pressure Profile Multiplier 1

Backflow Direction Specification Method **Normal to Boundary**

Backflow Pressure Specification Total Pressure

☐ Prevent Reverse Flow

☐ Radial Equilibrium Pressure Distribution

☐ Average Pressure Specification

☐ Target Mass Flow Rate

Apply Close Help

Setting zone id of part-copper to 5.
Setting zone id of wall-part-copper-part-water to 1.

PHYSICS

SETUP

C:\THE RIGTH MODEL Parallel Fluent@MININT-D5IGV1C.UP.AC.ZA [3d, dp, pbns, lam, 6-processes] [CFD Solver - Level 2]

File Domain Physics User-Defined Solution Results View Parallel Quick Se...

Mesh Zones Interfaces Mesh Models

Display... Info Units... Check Quality Scale... Transform Make Polyhedra Combine Separate Adjacency... Delete... Deactivate... Activate... Append Replace Mesh... Replace Zone...

Outline View Task Page

Filter Text

Setup

- General
- Models
- Materials
 - Fluid
 - Solid
- Cell Zone Conditions
- Boundary Conditions**
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- Reference Frames
- Named Expressions
- Curvilinear Coordinate System

Solution

- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Automatic Mesh Adaption
- Initialization
- Calculation Activities
- Run Calculation

Results

- Surfaces
- Graphics
- Plots
- Animations
- Reports

Parameters & Customization

Boundary Conditions

Zone Filter Text

- inlet
- interior-part-copper
- interior-part-water
- outer_surface
- outlet
- tube_face_wall**
- wall-part-copper-part-water
- wall-part-copper-part-water-shadow

Phase mixture Type wall ID 10

Edit... Copy... Profiles... Parameters... Display Mesh... Operating Conditions... Periodic Conditions... Perforated Walls...

Wall

Zone Name tube_face_wall

Adjacent Cell Zone part-copper

Momentum Thermal Radiation Species DPM Multiphase UDS Potential Structure Ablation

Thermal Conditions

- Heat Flux**
- Temperature
- Convection
- Radiation
- Mixed
- via System Coupling
- via Mapped Interface

Heat Flux [W/m²] 0

Wall Thickness [m] 0

Heat Generation Rate [W/m³] 0

Shell Conduction 1 Layer Edit...

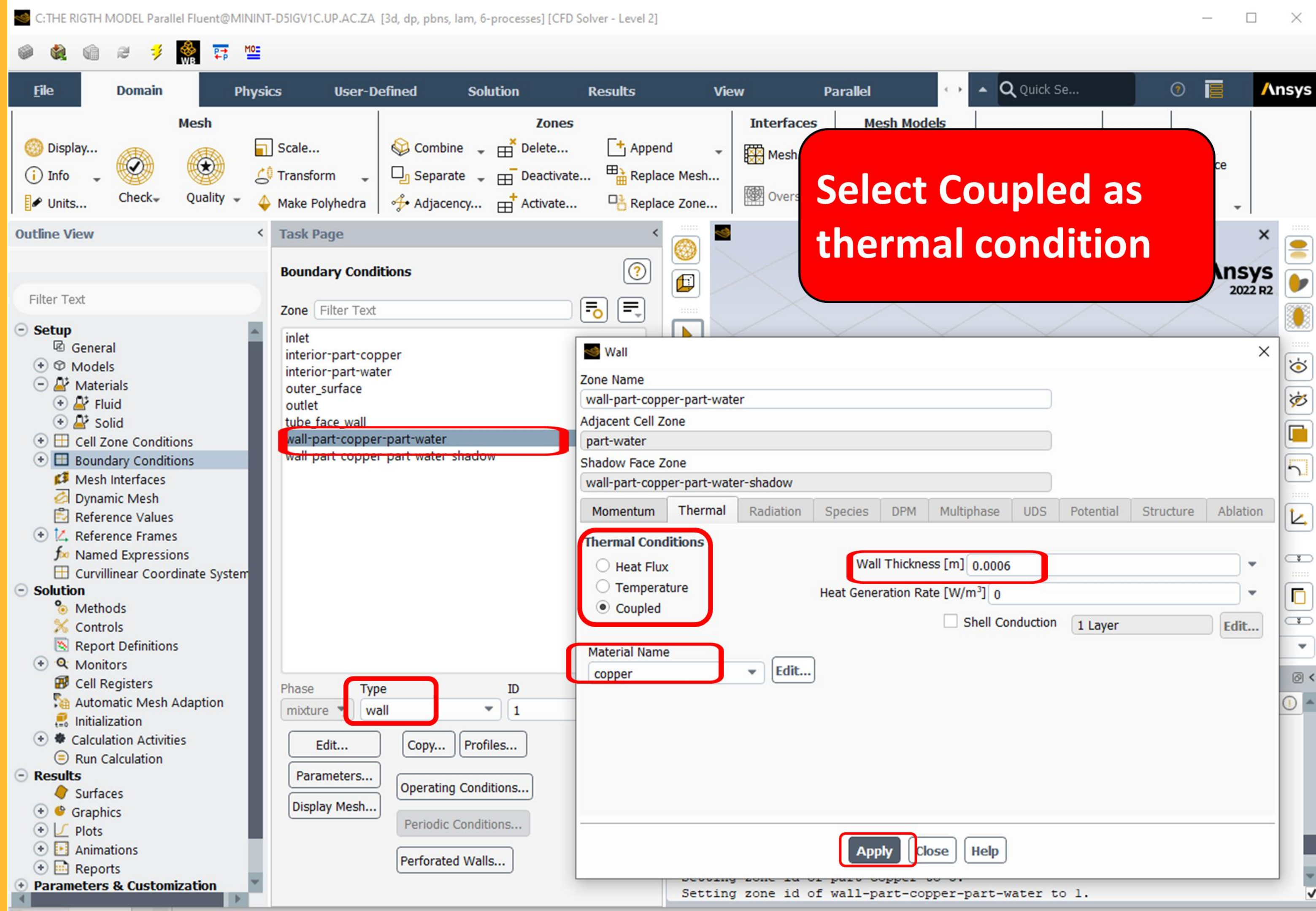
Material Name copper Edit...

Apply Close Help

Setting zone id of wall-part-copper-part-water to 1.

PHYSICS

SETUP



PHYSICS

SETUP

The image shows the ANSYS Fluent interface. On the left is the 'Outline View' panel, and on the right is the 'Task Page' panel.

Outline View:

- Setup
 - General
 - Models
 - Materials
 - Fluid
 - Solid
 - Cell Zone Conditions
 - Fluid
 - Solid
 - Boundary Conditions
 - Inlet
 - Internal
 - Outlet
 - Wall
 - Mesh Interfaces
 - Dynamic Mesh
 - Reference Values
 - Reference Frames
 - Named Expressions
 - Curvilinear Coordinate System
- Solution** (highlighted with a red box)
 - Methods** (highlighted with a red box)
 - Controls
 - Report Definitions
 - Monitors
 - Cell Registers
 - Automatic Mesh Adaption
 - Initialization
 - Calculation Activities
 - Run Calculation
- Results
 - Surfaces
 - Graphics
 - Plots

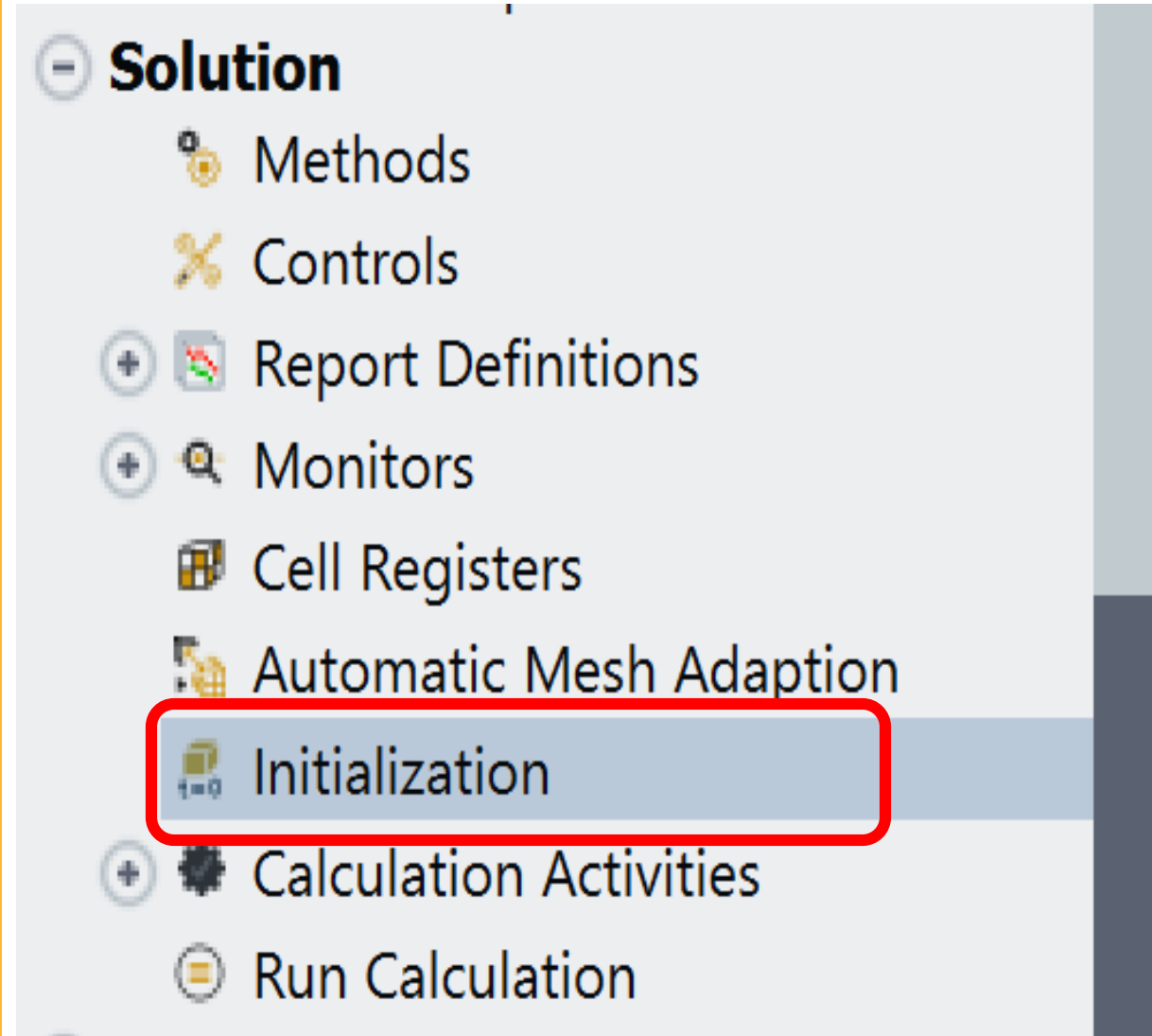
Task Page:

- Solution Methods**
 - Pressure-Velocity Coupling**
 - Scheme** (highlighted with a red box): SIMPLE
 - Flux Type: Rhie-Chow: momentum based ☒ Auto Select
 - Spatial Discretization**
 - Gradient: Least Squares Cell Based
 - Pressure: Second Order
 - Momentum: Second Order Upwind
 - Energy: Second Order Upwind
 - Pseudo Time Method: Off
 - Transient Formulation: (dropdown menu)

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

PHYSICS

SETUP



Solution Initialization

Initialization Methods

- ☐ Hybrid Initialization
- ☒ Standard Initialization

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



PHYSICS

SETUP

Outline View

Filter Text

- + Solid
- Cell Zone Conditions
 - + Fluid
 - + Solid
- Boundary Conditions
 - + Inlet
 - + Internal
 - + Outlet
 - + Wall
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- + Reference Frames
- Named Expressions
- Curvilinear Coordinate System
- **Solution**
 - Methods
 - Controls
 - Report Definitions
 - + Monitors
 - Cell Registers
 - Automatic Mesh Adaption
 - Initialization
 - + Calculation Activities
 - Run Calculation**
- Results
 - Surfaces
 - + Graphics
 - + Plots
 - + Animations
 - + Reports
- + Parameters & Customization
- + Simulation Reports

Task Page

Run Calculation

Check Case... Update Dynamic Mesh...

Parameters

Number of Iterations: 10000 Reporting Interval: 1

Profile Update Interval: 1

Solution Processing

Statistics

☐ Data Sampling for Steady Statistics

Data File Quantities...

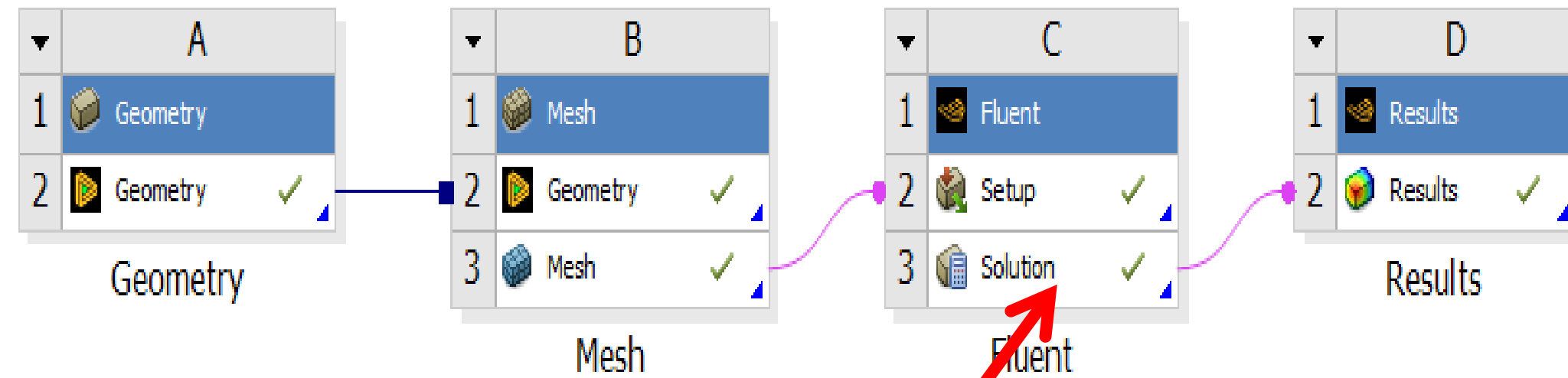
Solution Advancement

Calculate

**Set Number of Iterations
As 10,000 and Click
"Calculate"**

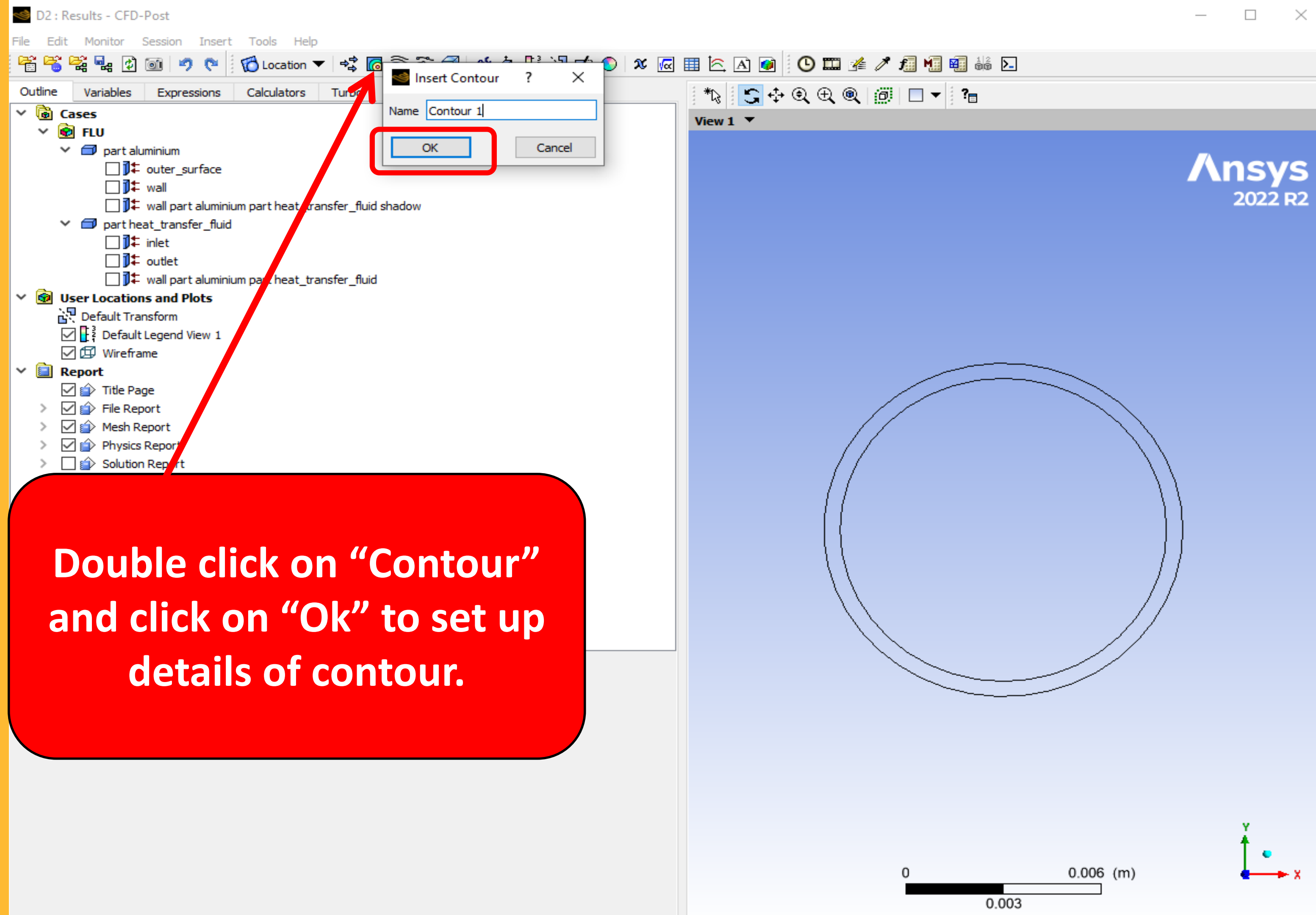
POST PROCESSING RESULTS

- Mechanical Model
- Mesh
- Microsoft Office Excel
- Performance Map
- Polyflow
- Results**
- System Coupling
- Turbo Setup
- TurboGrid

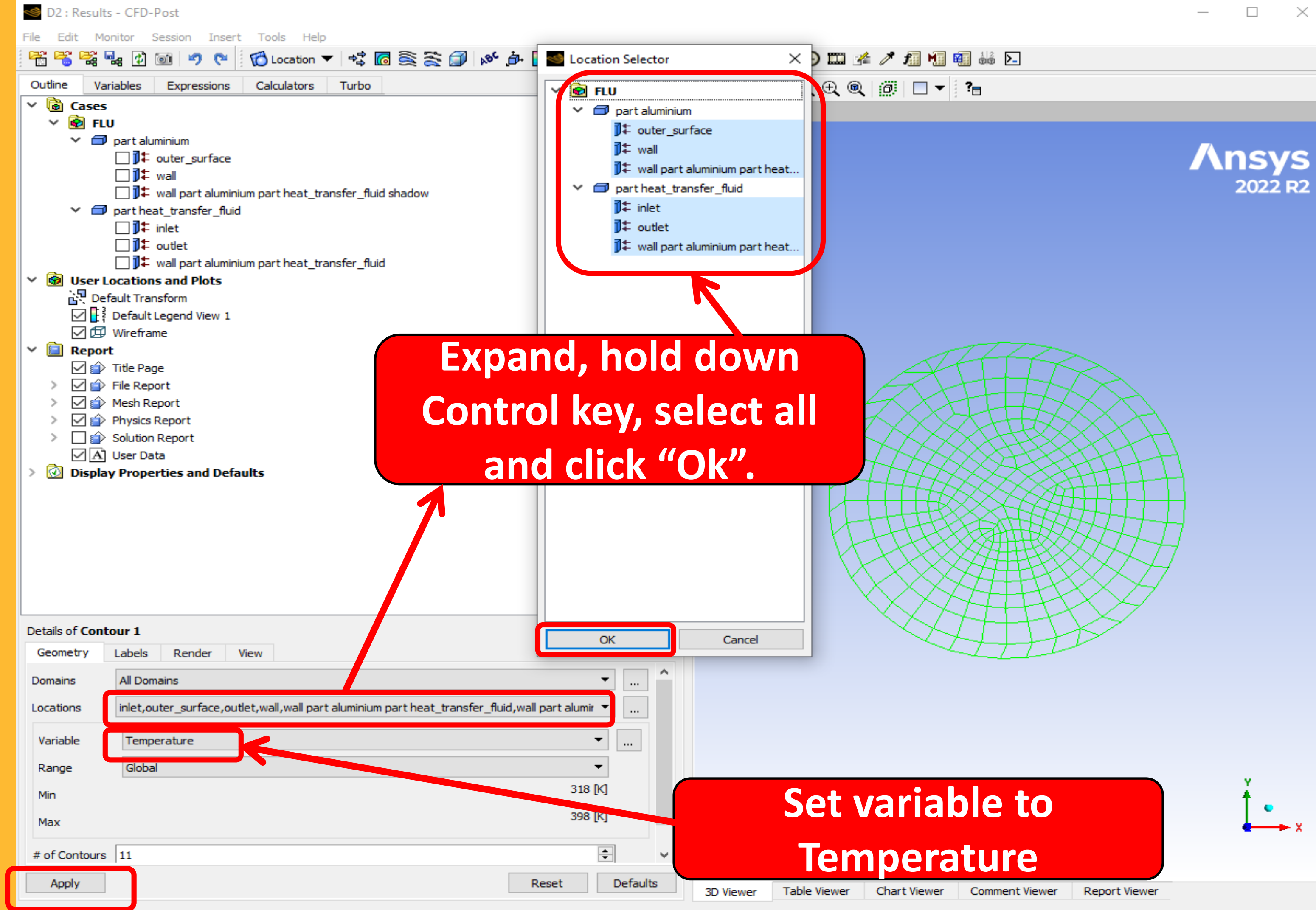


**Drag "Result" to
"Solution" and drop.**

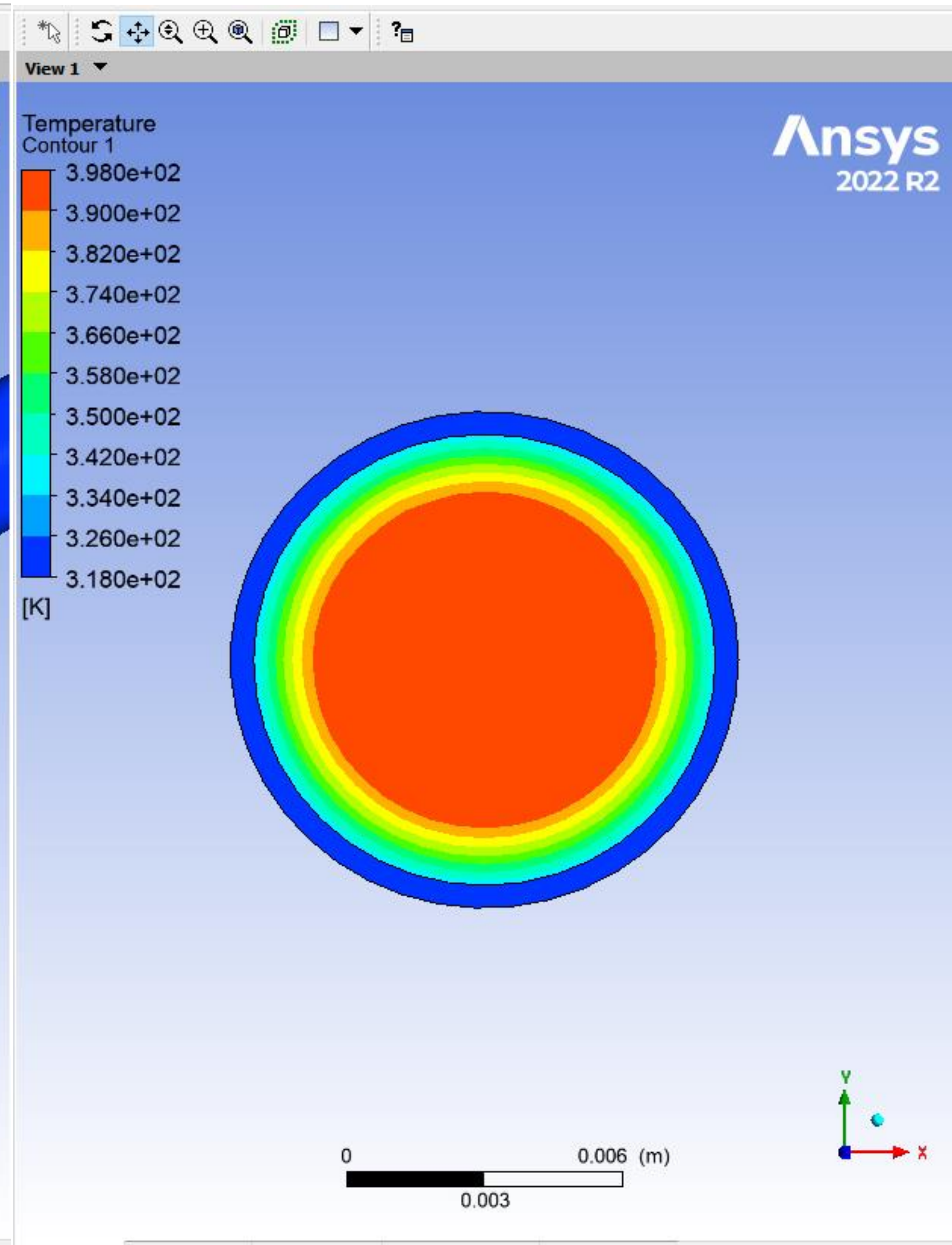
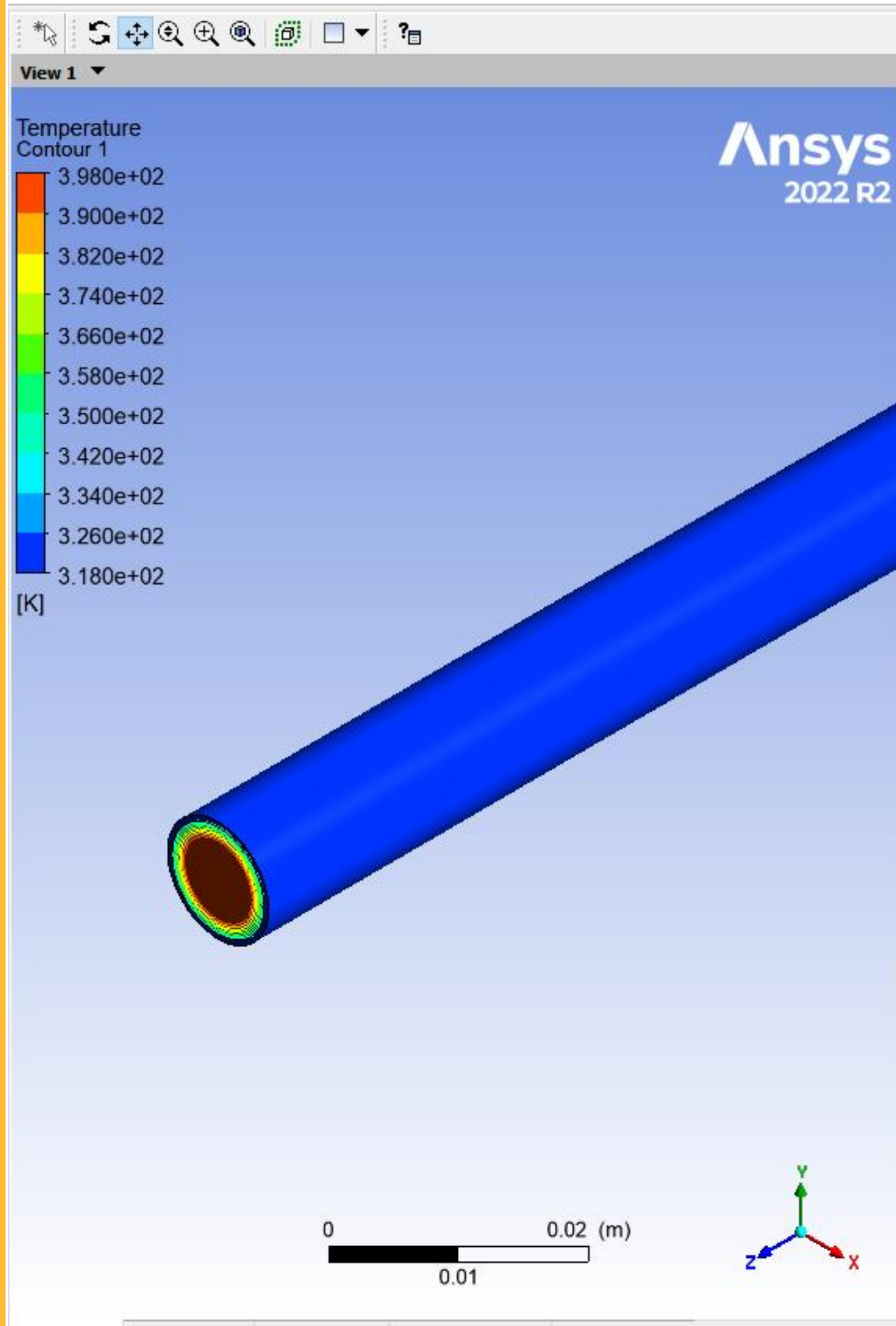
POST PROCESSING



POST PROCESSING



POST PROCESSING



POST PROCESSING

D2 : Uniform heat flux - CFD-Post

File Edit Monitor Session Insert Tools Help

Outline Variables Expressions Calculators Turbo

Cases

- Copy of Fluent
 - part copper
 - face_wall
 - heat_flux_wall
 - wall part copper part water shadow
 - part water
 - inlet
 - outlet
 - wall part copper part water
- User Locations and Plots**
 - Contour 1
 - Contour 2
 - Default Transform
 - Default Legend View 1
 - Line 1
 - Line 2**
 - Plane 1
 - Wireframe
- Report**
 - Title Page
 - File Report
 - Mesh Report
 - Physics Report
 - Solution Report
 - User Data

Details of Line 2

Geometry Color Render View

Domains: All Domains

Definition

Method: Two Points

Point 1: 0 0 0.5

Point 2: 0 0.00255 0.5

Line Type: ☐ Cut ☒ Sample

Apply Reset Defaults

View 1

Ansys 2022 R2

Insert Line location(Line 1) along tube radius

Set details of line and apply

POST PROCESSING

Insert Chart

Select line 1 for Data Series

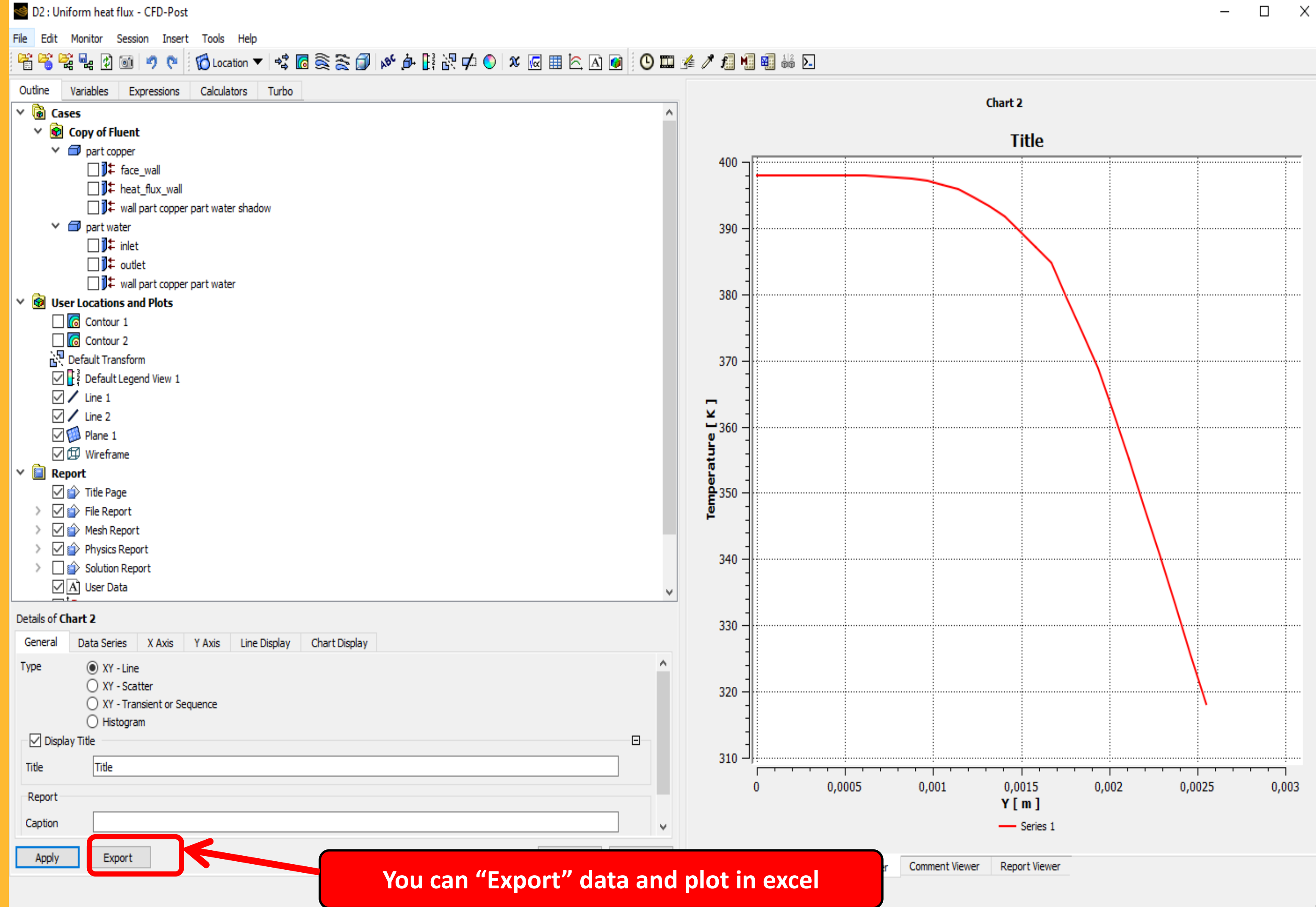
Set Variable as Y for X Axis

Set Variable as Temperature for Y Axis

Apply

The image shows the ANSYS CFD-Post software interface. The left sidebar contains a tree view with categories like Cases, User Locations and Plots, Report, and Display Properties and Defaults. The 'Insert' menu is open, showing options like Vector, Contour, Streamline, Particle Track, Volume Rendering, Text, Coordinate Frame, Legend, Instance Transform, Clip Plane, Color Map, Variable, Expression, Table, Chart, Comment, and Figure. The 'Chart' option is highlighted. The 'Details of Chart 1' panel is open, showing the 'Data Series' tab with 'Line 1' selected. The 'X Axis' tab is also open, showing 'Variable' set to 'Y'. The 'Y Axis' tab is open, showing 'Variable' set to 'Temperature'. The 'Apply' button is highlighted.

POST PROCESSING



POST PROCESSING

D2 : Uniform heat flux - CFD-Post

File Edit Monitor Session Insert Tools Help

Outline Variables Expressions **Calculators** Turbo

Macro Calculator
Mesh Calculator
Function Calculator

**Set function as mass flow ave
Set location as outlet
Set variable as Temperature**

Function Calculator

Function	massFlowAve
Location	outlet
Case	Copy of Fluent
Variable	Temperature
Direction	None

Results

Mass Flow Average of Temperature on outlet

375.601 [K]

Required Outlet Temperature

☒ Clear previous results on calculate
☐ Show equivalent expression

Calculate Hybrid Conservative

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**