Electronics Cooling with Natural Convection and Radiation

- Introduction to ANSYS Fluent

Department of Innovative Design and Integrated Manufacturing Lab.
Mechanical and Aerospace Engineering
Seoul National University

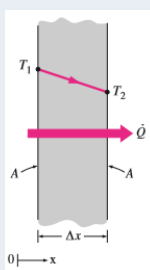
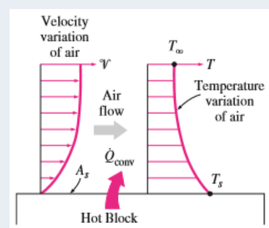
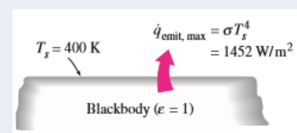
Nov 7th 2017

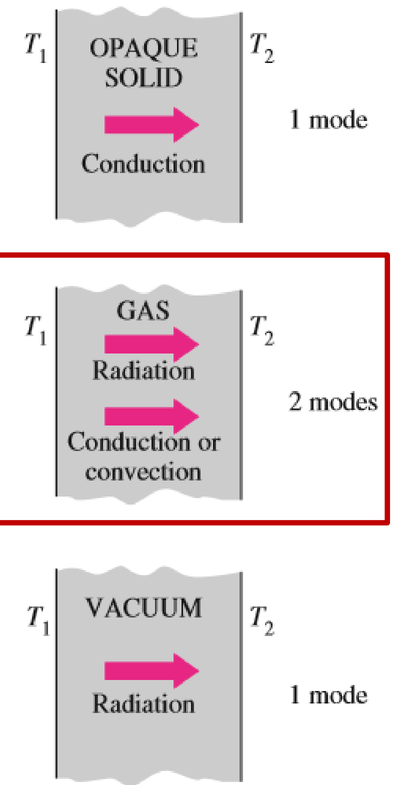
- Example
  - In this example, analysis will be performed on heatsink heat dissipation which is generated from PCB (Printed Circuit Board).

- Contents
  - This example contains the following:
    - Heat conduction through solid material
    - Natural convection of hot air
    - Appropriate heat transfer
  - In this class, we will skip meshing method.

- Objectives
  - Learn how to perform heat transfer analysis with CFD.

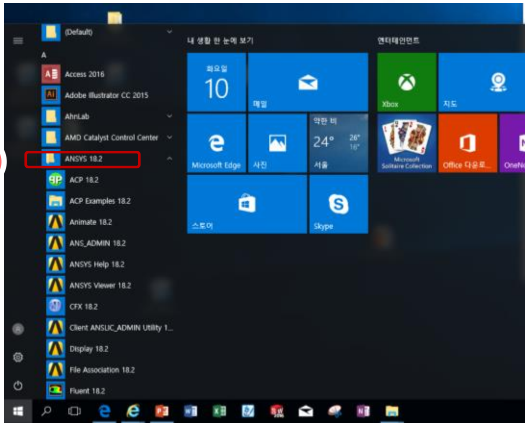
- Heat transfer concept
  - In this example, air will be used as a medium.
  - Radiation model and conduction/convection model is needed this “Heatsink” analysis.

	Conduction	Convection	Radiation
Coefficient	**K (Thermal conductivity)**	**H (Convection heat transfer coefficient)**	**ε (Emissivity) σ (Stefan-Boltzmann constant)**
Diagram			
Equation	$\dot{Q}_{cond} = -kA\frac{dT}{dx}$	$\dot{Q}_{conv} = hA_s(T_s - T_\infty)$	$\dot{Q}_{rad} = \varepsilon\sigma A_s\,(T_s^4 - T_{(surr)}^4)$



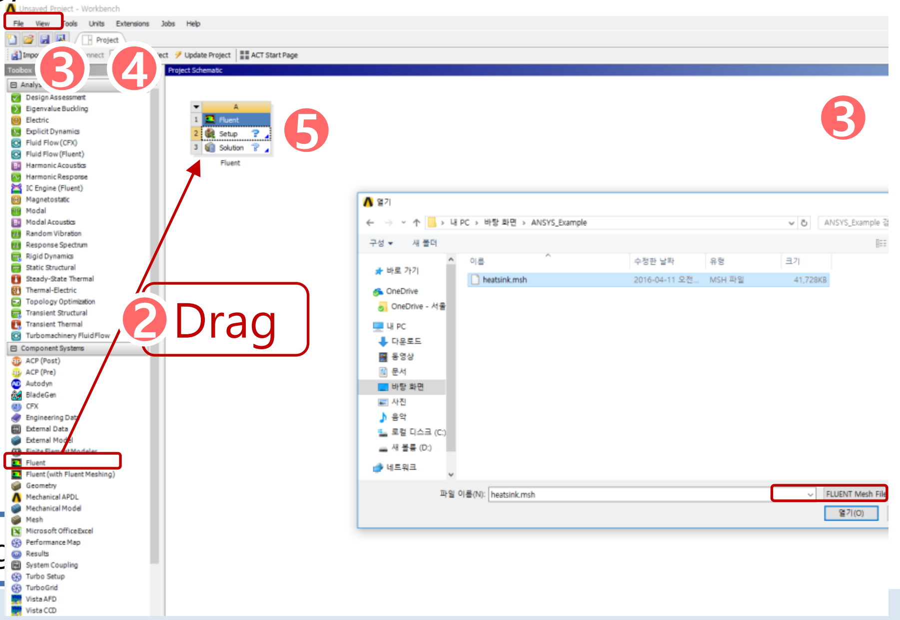
HEAT TRANSFER, A Practical Approach, SECOND EDITION, YUNUS A. CENGEL. pp. 30

- Start Fluent
  1. Start → ANSYS 18.2 → Workbench
  2. Form the Workbench, start Fluent

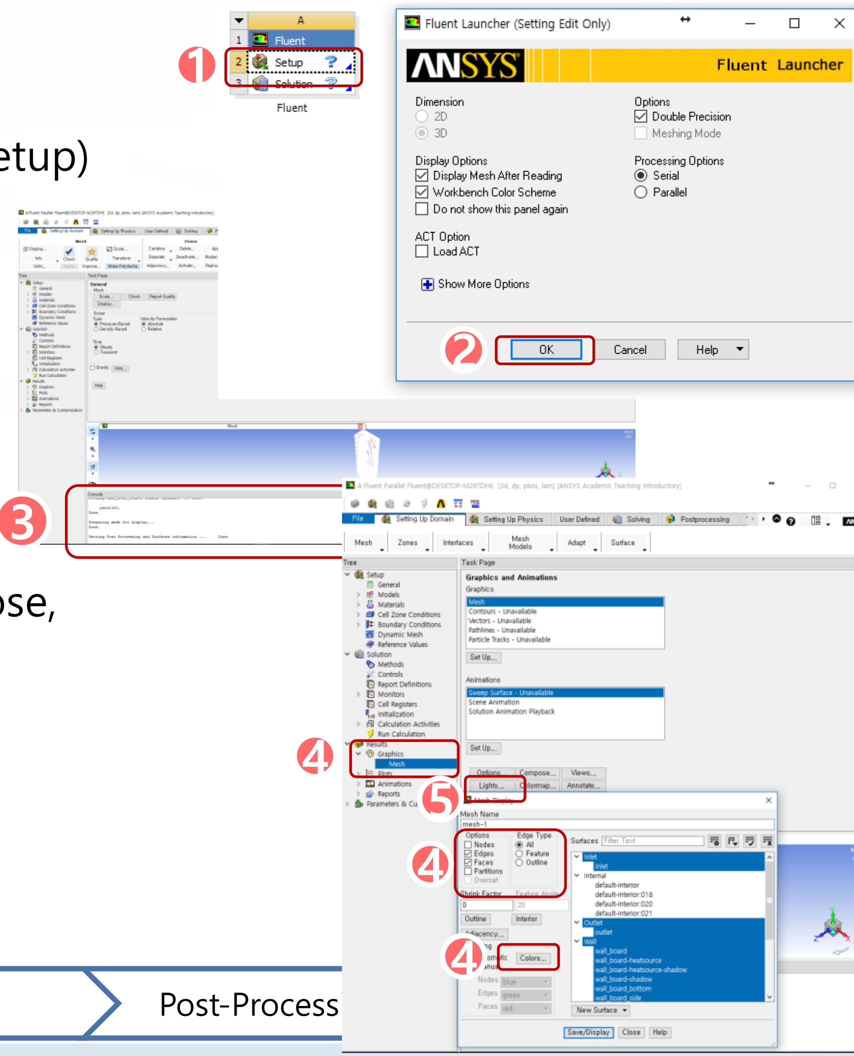


- Fluent setting
  3. View → Properties → Set the Double Precision (Better convergence, strict energy balance)
  4. File → Save As “Electronics_Cooling”

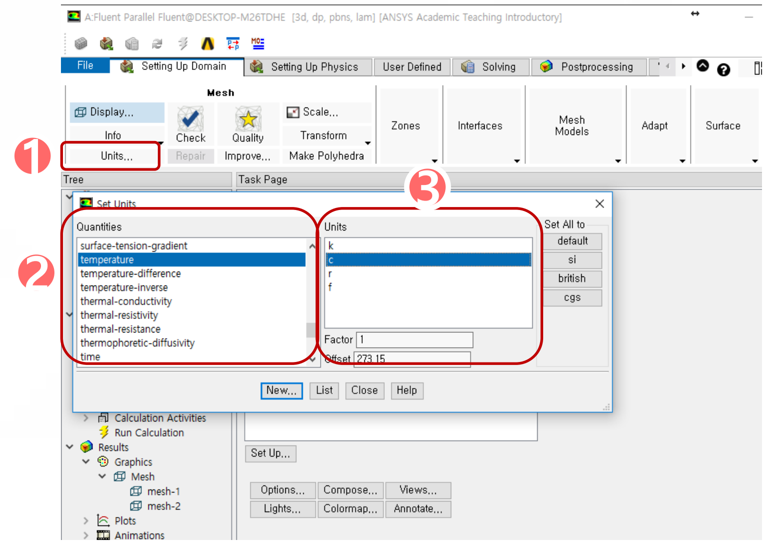
- Load Mesh file
  5. On the setup menu click right Button of the mouse
     – Click “Import FLUENT Case”
  6. Change file format to Fluent Mesh File
     – Find and click Heatsink.msh
     – Open

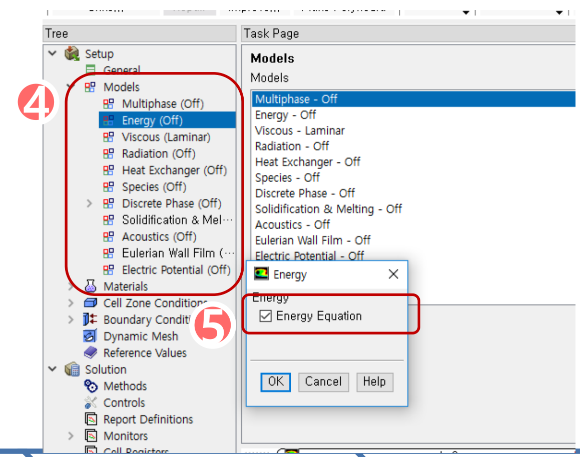


- Initiate the Fluent
  1. Run with Default option (Double click setup)
  2. OK
- View the model
  3. Check the TUI, if there is no error.
  4. Mesh display and Display setting
     - Graphics → Mesh
     - Face (deactivate), Edge type → Feature
     - Colors click → Color by ID check → Close,
     - Display click
  5. Lights → Headlight on activate



- Temperature unit change K → °C
  1. User-Defined → Units
  2. Quantity → Temperature
  3. Temperature units “°C”
  - Close
- Energy Equation activate
  - Tree → Models → Energy double click
  - Energy check (activate) → ok

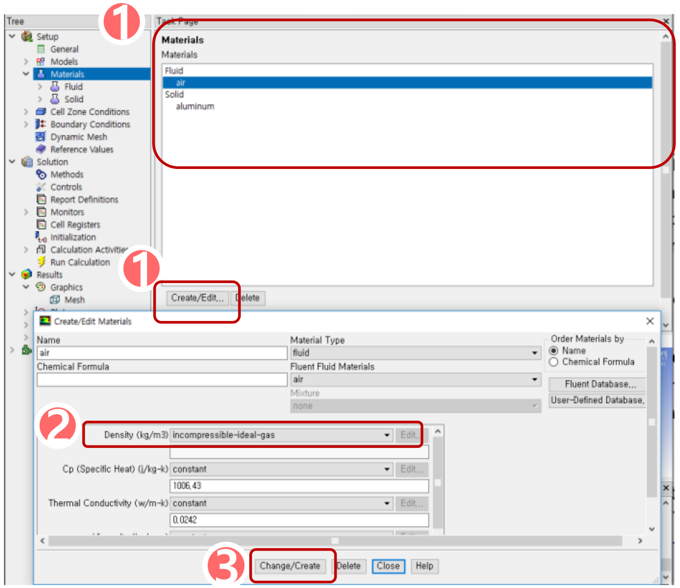




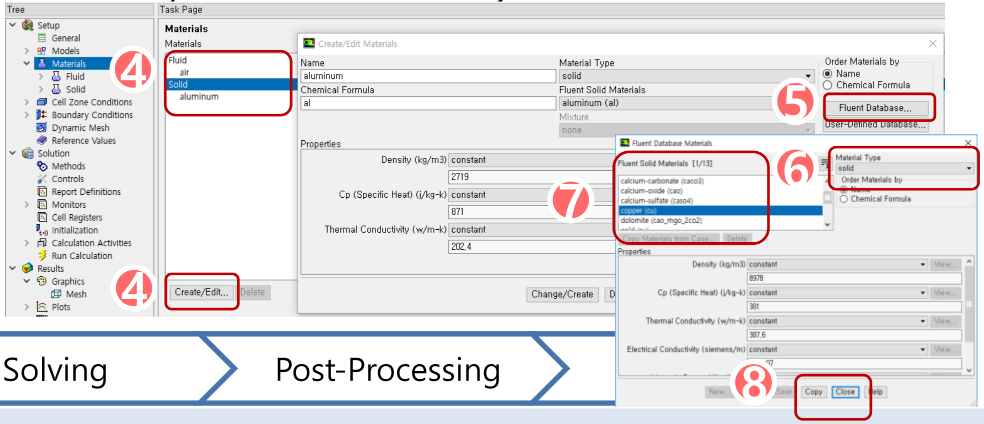
- Comments on Model setup
  - Check the mesh display, if there is any missing.
  - Check the unit. Workbench uses SI-Units (meter, kg etc.)
  - In many cases, on isothermal condition, we don’t have to use energy equation because of CPU load. But in this case, we activate energy equation.

Material Properties

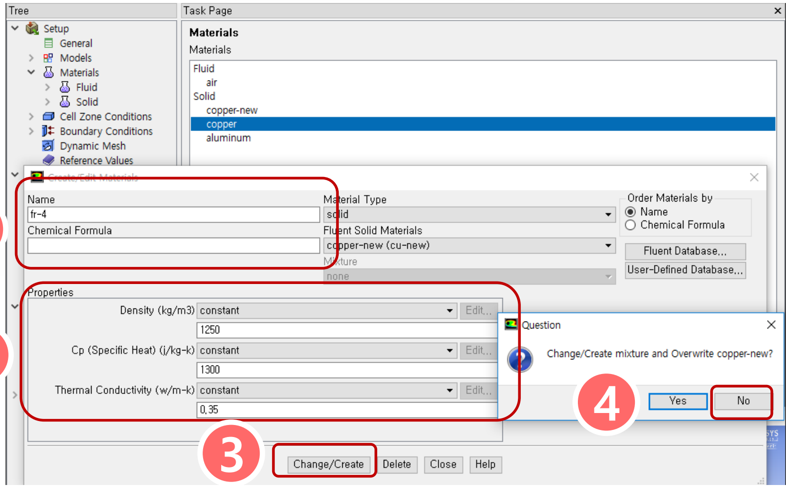
- Air density changes with thermal condition
  1. Materials → Air → Create/Edit
  2. Density → Incompressible ideal gas
     – Other properties remain default
  3. Change/Create click
     – Create/Edit Materials close



- Add solid materials (Board, Heat Sink, Heat Source)
  4. Materials → Solid → Create/Edit
  5. "Fluent Database" click
  6. Type → Solid
  7. Select "Copper"
  8. Copy → Close



- Put the PCB properties into the copper properties
  1. Select “copper” → PCB is consisted of “fr-4”
  – Change the name to “fr-4”
  – Delete chemical formula
  2. Density = $1250 kg/m^3$, cp = $1300\ J/KG \cdot K$ conductivity $0.35\ W/m \cdot K$
  3. Change/Create click
  4. To not overwrite to copper select “No”

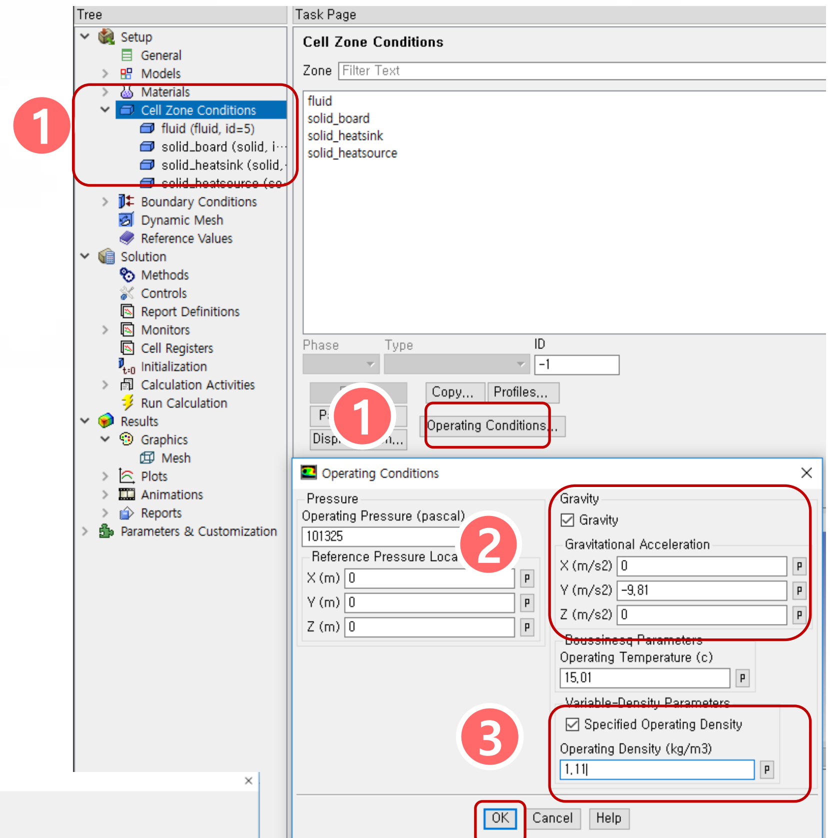


- Same as the above, add “Component” Material
  – Density = $1900\ kg/m^3$, cp =$795\ J/kg \cdot K$, Thermal conductivity = $10\ W/m \cdot K$

- Comments on Material properties
  - For natural convection problems, flow generate due to difference between density. In the simulation region we use incompressible ideal gas instead of compressible ideal gas because pressure change influence is minimal.
  - Although fluent database have default property of materials, consisted of reference STP(0°C,1atm)/RTP(25°C,1atm) values, it needs to be checked before starting analysis.

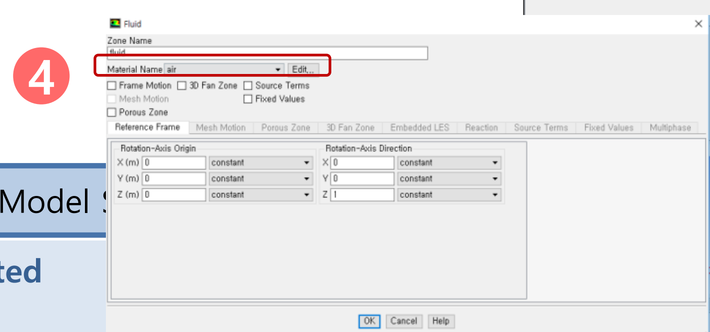
Set Cell Zone Conditions

1. Cell Zone Conditions
→ Click "Operating Conditions"
2. Activate "Gravity" → put y-direction gravity value "$-9.81 m/s^2$"
3. Activate "Specified Operating Density" and put 1.11 kg/m^3
– Click "OK"



- Don't change fluid zone

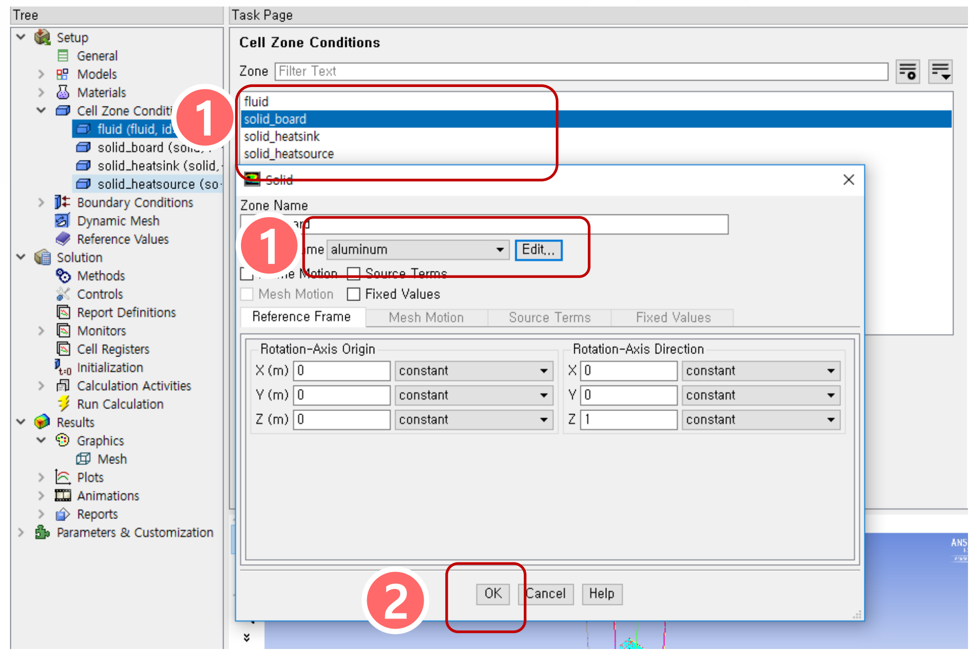
4. Cell Zone → Fluid
→ Check if the Material is "Air"
– Close

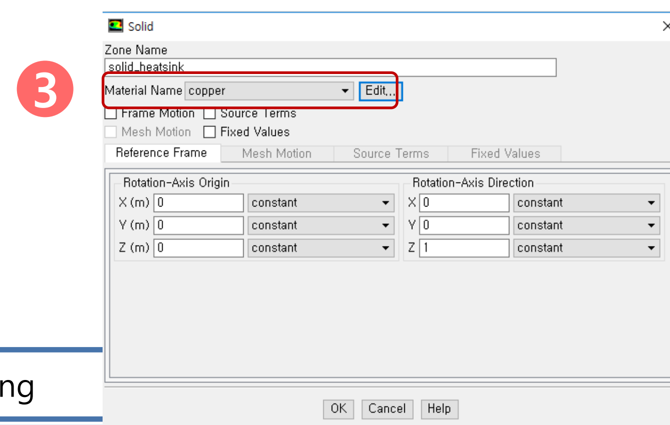


Set Cell Zone Conditions

- PCB properties
  1. Cell Zone → solid_board
     → Material change to "fr-4" → Edit
  2. Click "OK"

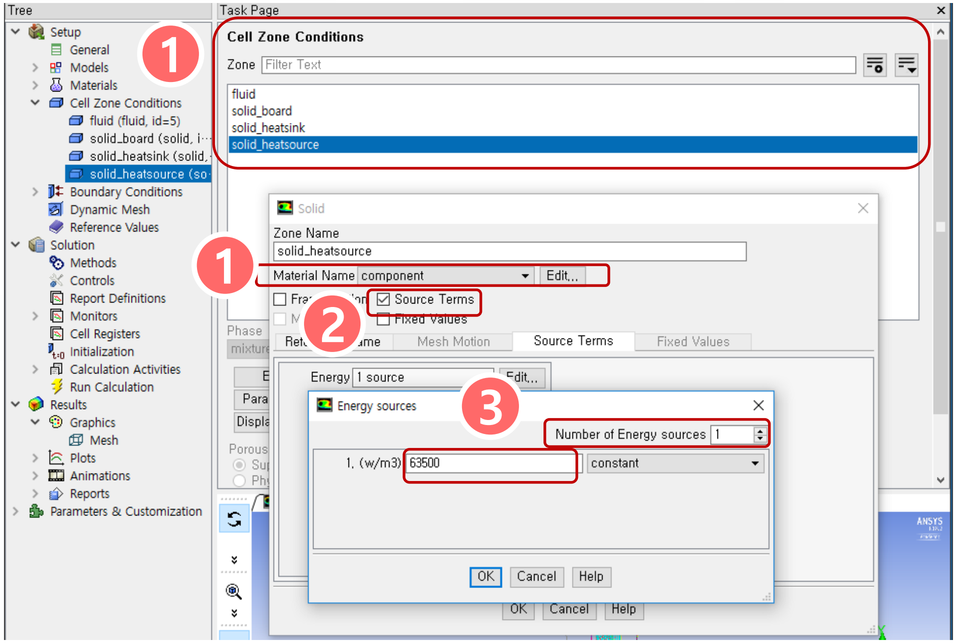
- Heatsink property
  3. Cell Zone → solid_heatsink
     – Material change to "Copper"
     – Click "Edit" and check the values





Set Cell Zone Conditions

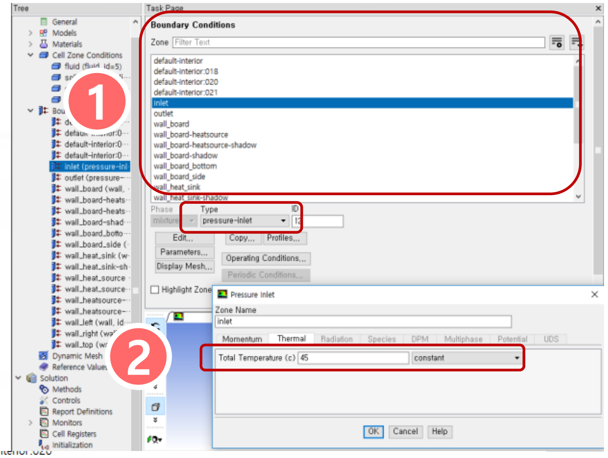
- Condition on "Component"
  1. Cell Zone → solid_heatsource
  → Material change to "component"
  2. Activate "Source Terms"
  → Move to "Source Terms tap" → Edit
  3. "Number of Energy Sources" 1
  → constant
  → put volume energy "635,000 W/m^3"
  (Material Heat Dissipation 75W/ Volume)

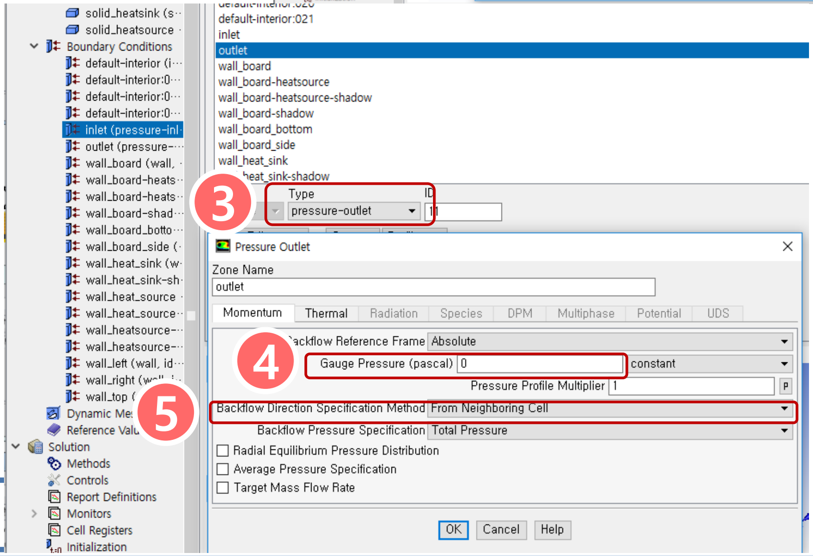


Set Boundary Conditions

- Inlet condition (Boundary Conditions < Inlet):
  1. Pressure Inlet
  2. Thermal tap → set 45 °C → "OK"

- Outlet condition
  3. Maintain "Pressure Outlet"
  4. Set gauge pressure 0 Pa
  5. Direction "From Neighboring Cell"
  6. "Thermal tap" → "Backflow Total-Temperature" Set to 45 °C → "OK"



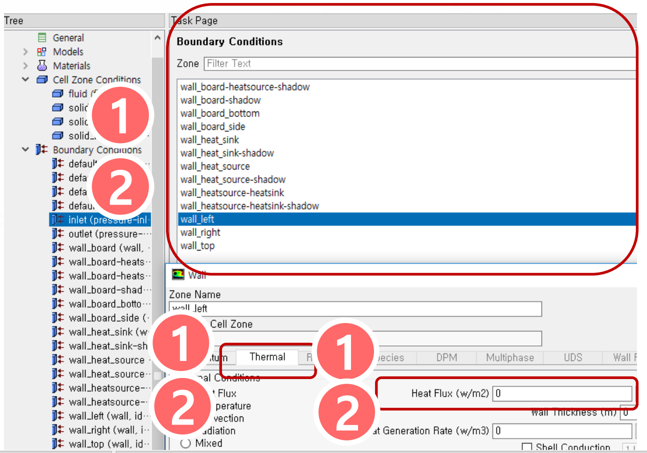


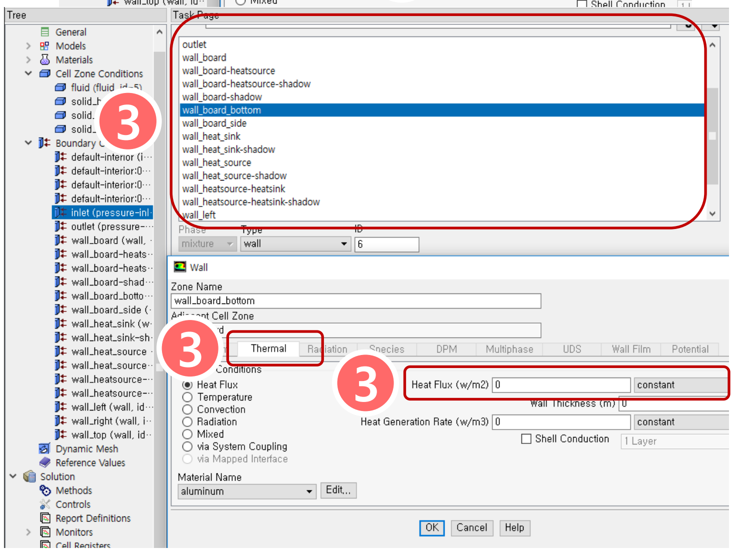
Set Boundary Conditions

- Adiabatic wall condition
  1. Boundary zone → select "wall_left"
     – Thermal tab → check "heat flux" 0
  2. Set the "wall_right" and "wall_top" same as above

- Select PCB's surface "adiabatic"
  3. Surface → wall_board_bottom → "Heat Flux" 0
     – Set the "wall_board_side" same as above

- Wall_board is PCB's fluid region not an outer boundary





Introduction | Basic Model Setup | Solving | Post-Processing | Summary

Comments on Cell and Zone Conditions

- Operating conditions
  - For calculating the natural convection, gravity condition is needed. (But in many cases, gravity effect can be neglected.)
  - Operating density is very important in natural convection problem. We use density condition from the "Far Field Temperature" value. (Ex, Inlet temperature)
  - Operating pressure position is related to the inlet/outlet pressure condition.

- Boundary zones
  - Inlet and outlet pressure will be 0 Pa because it is on the operating pressure region.

Comments on Cell and Zone Conditions

- Volume data
  - On volume integrals panel, we can check the volume. However, before activating volume integral, initiation process is needed.

- External heat loss
  - In this problem, adiabatic condition of the outer wall is needed.
  - When it is needed, we can modify the other FLUENT condition.
    - Heat flux condition
    - Heat transfer coefficient condition (Ex. Vertical planar natural convection)
    - Radiation heat transfer (emissivity and temperature, regardless of the radiation model in FLUENT)
    - Shell conduction (conduction analysis considering material and thickness)

Comments on Cell and Zone Conditions

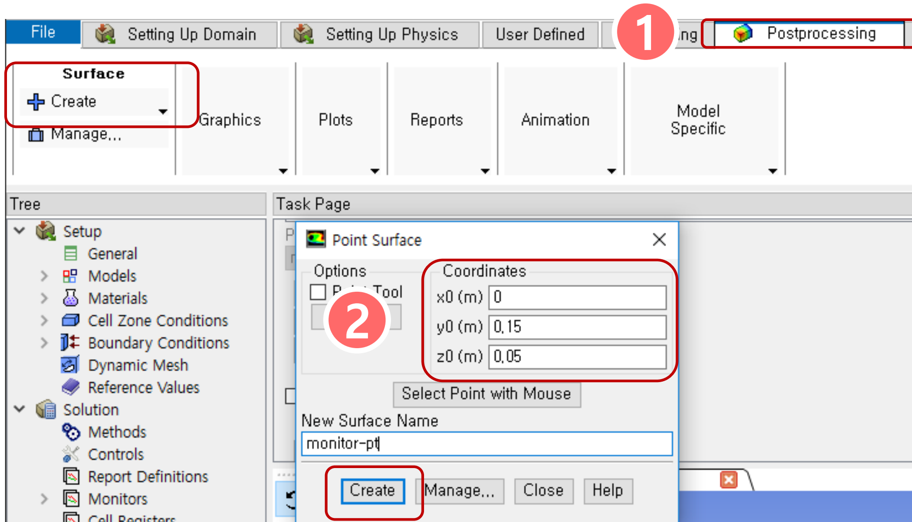
- Internal boundary condition
  - 4 cell zones exist (2 fluid and 2 solid_board)
  - For dividing 2 zones we use pre-processing for making wall_board
  - To bring mesh, FLUENT makes shadow surface : wall_board_shadow

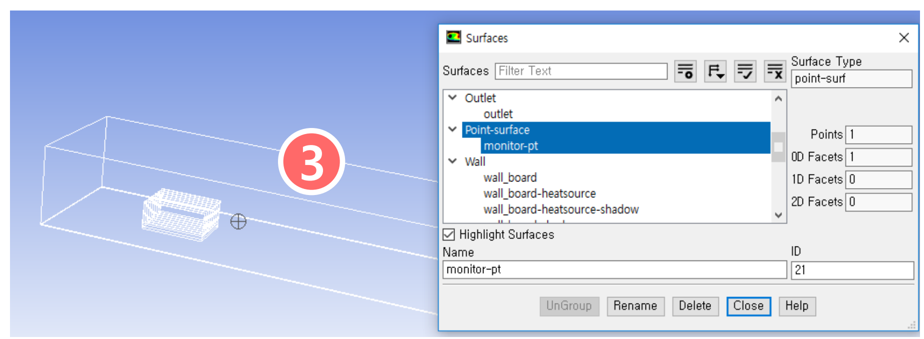
- Opening the two boundary condition
  - wall_board has the adjacent cell zone solid_board
  - wall_board_shadow has the adjacent cell zone fluid

- Check the surface if emissivity and wall roughness are applied correctly

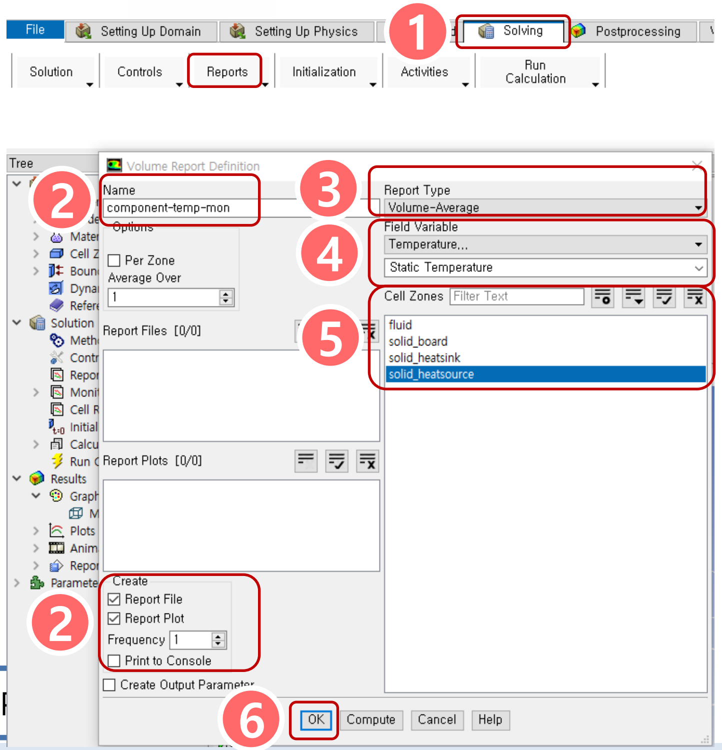
Create a Monitoring Point

- We can monitor air temperature by generating a Point Surface near the upper component
  1. Postprocessing → Surface → Create → Point
  2. Coordinates (in meters) put (0,0.15,0.05) and create
  3. Setting up domain → Surface(Manage) → Point (Position check)



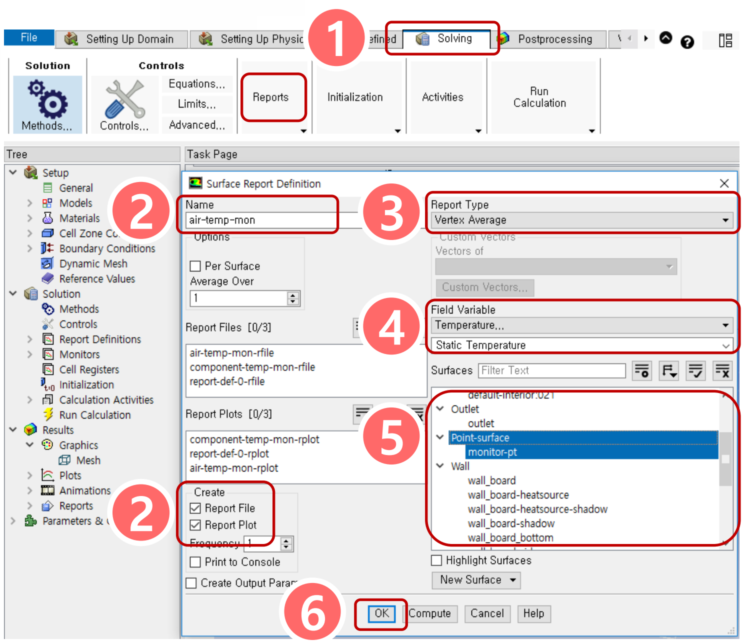


- Surface Monitor
    1. Ribbon tap → Solving → Reports → Definitions → New → Volume Report → Volume Average
    2. On name, put "component-temp-mon" → on "Create" → check Report File and Report Plot
    3. Report type → Volume-Average
    4. Variable → Temperature → Static Temperature
    5. Cell Zones → select "solid_heatsource"
    6. OK

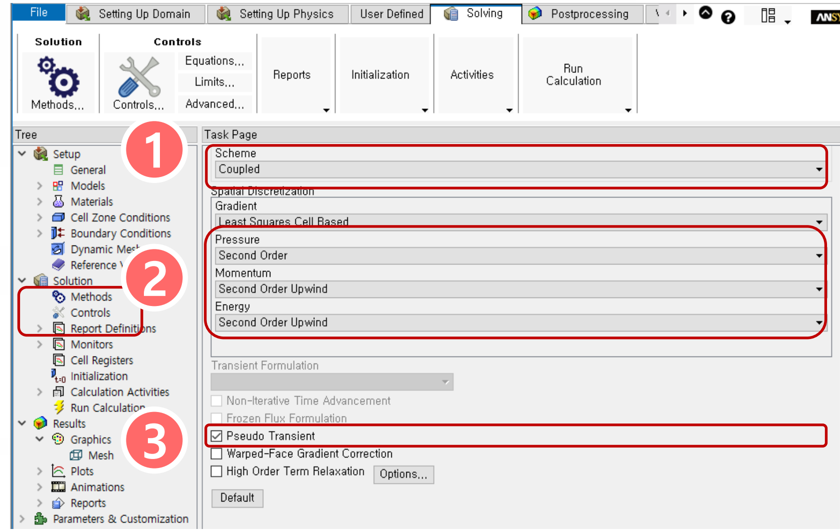


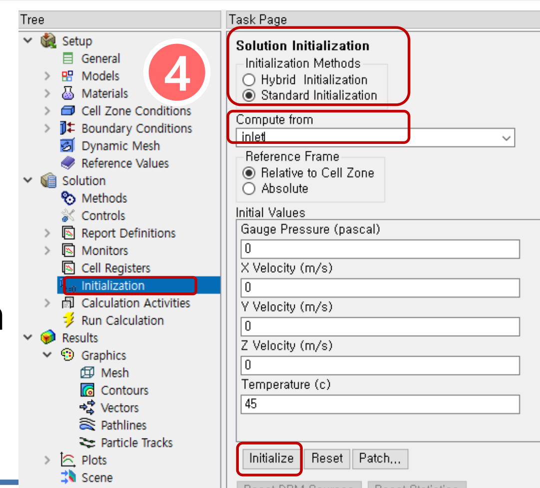
Fluent can only monitor temperatures in Kelvin or Rankine. This applies only to the surface monitor and does not affect the units you have defined elsewhere.

- Surface Monitor
  1. Ribbon tap → Solving → Reports → Definitions → New → Surface Report → Vertex Average
  2. On “name”, put “air-temp-mon”→ on “Create”→ check Report File and Report Plot
  3. Report type → Vertex Average
  4. Variable → Tmperature → Static Temperature
  5. Surfaces → select “monitor-pt”
  6. OK



- Solution
  1. Solution → Methods
  - On “Pressure-Velocity Coupling”
  - Select “Coupled” on “Scheme”
  2. On “Spatial Discretization”
  → Pressure → Body Force Weighted
  - Momentum and Energy
  → “2nd Order Upwind”
  3. Activate “Pseudo Transient”
  4. Solution → Initialization
     - Standard Initialization → Compute from → Inlet
     - Initialize
  5. File → Save Project (Workbench) or Write Case and Data





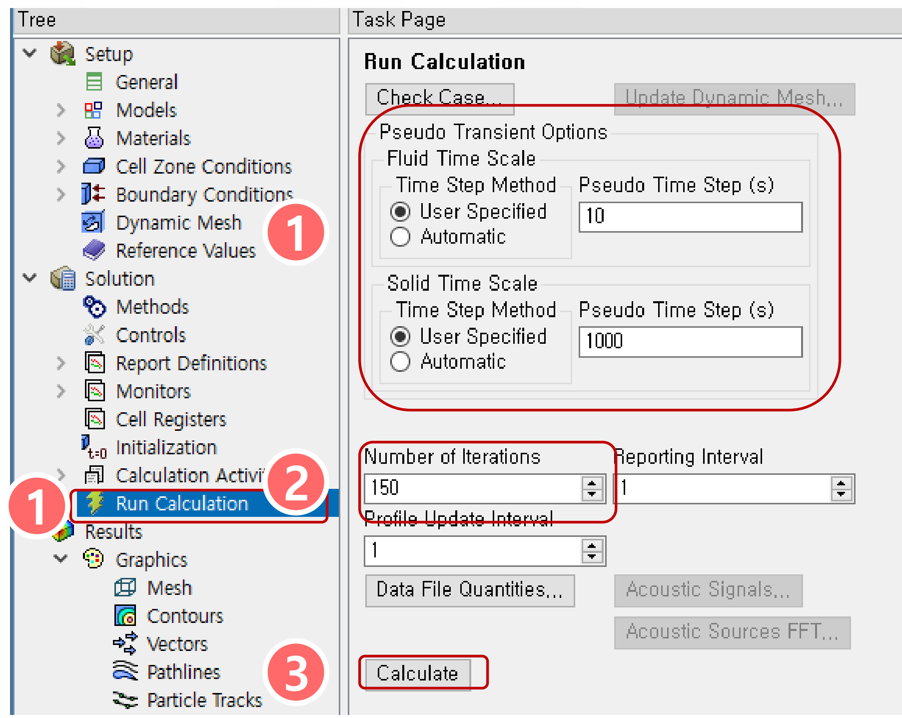
- Solution

  1. Run Calculation
     - On pseudo transient options → "Fluid Time Scale", "Solid Time Scale" → user Specified
     - Fluid 10s, Solid 1000s

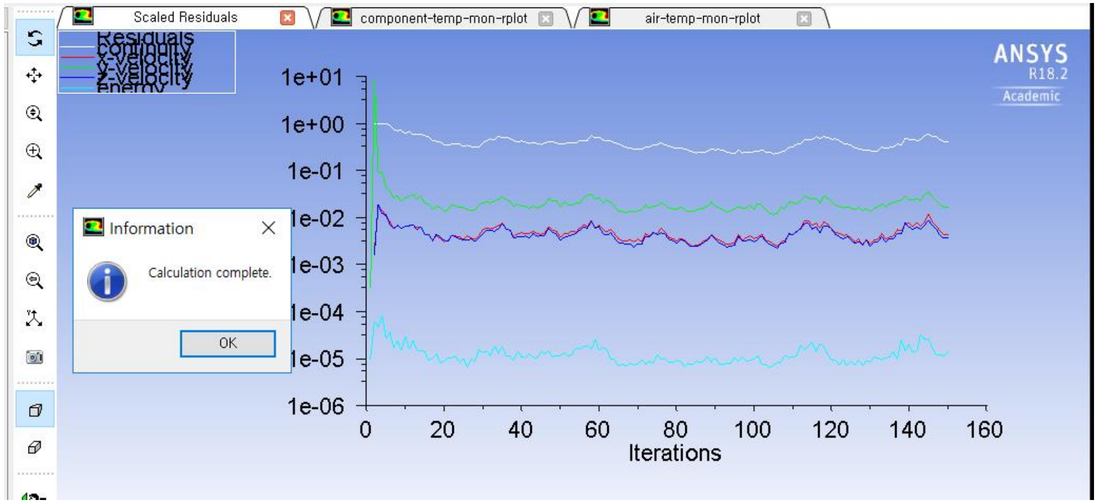
  – Now the preparation process is done.

  – It takes 5~10 minutes to converge.

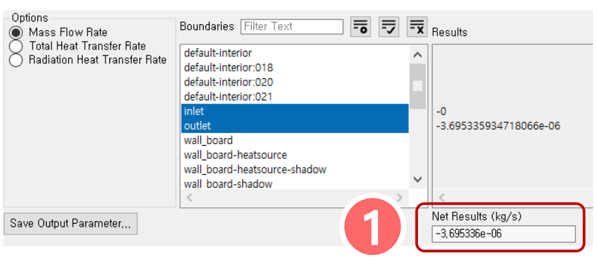
  2. Iteration : 150
  3. Run Calculation

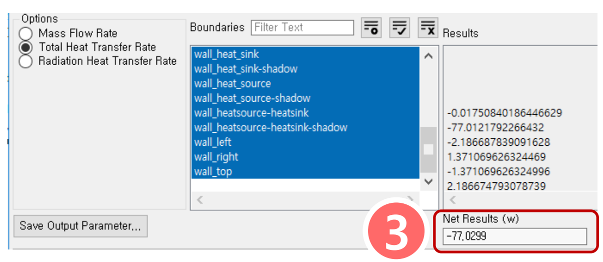


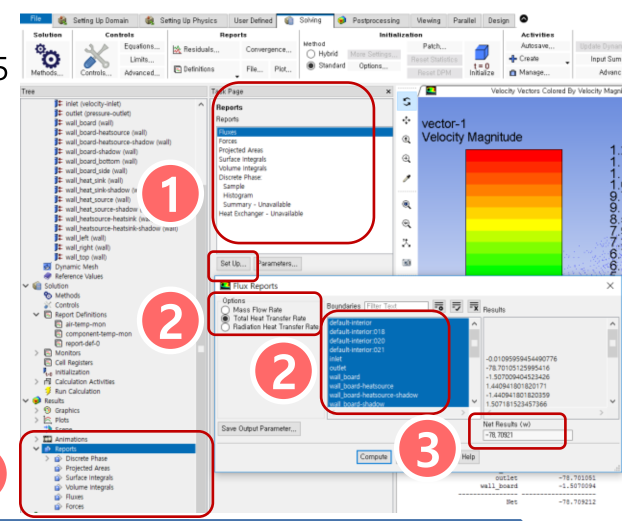
- Comments on Solver Controls
  - Solver setting is set for stable convergence.
  - This basic model setup is needed for applying Natural Convection Pressure Scheme and Body Force Weighted.
  - Solution initialization is used for initial iteration, and convergence speed will be faster if we put the value close to the converged value.



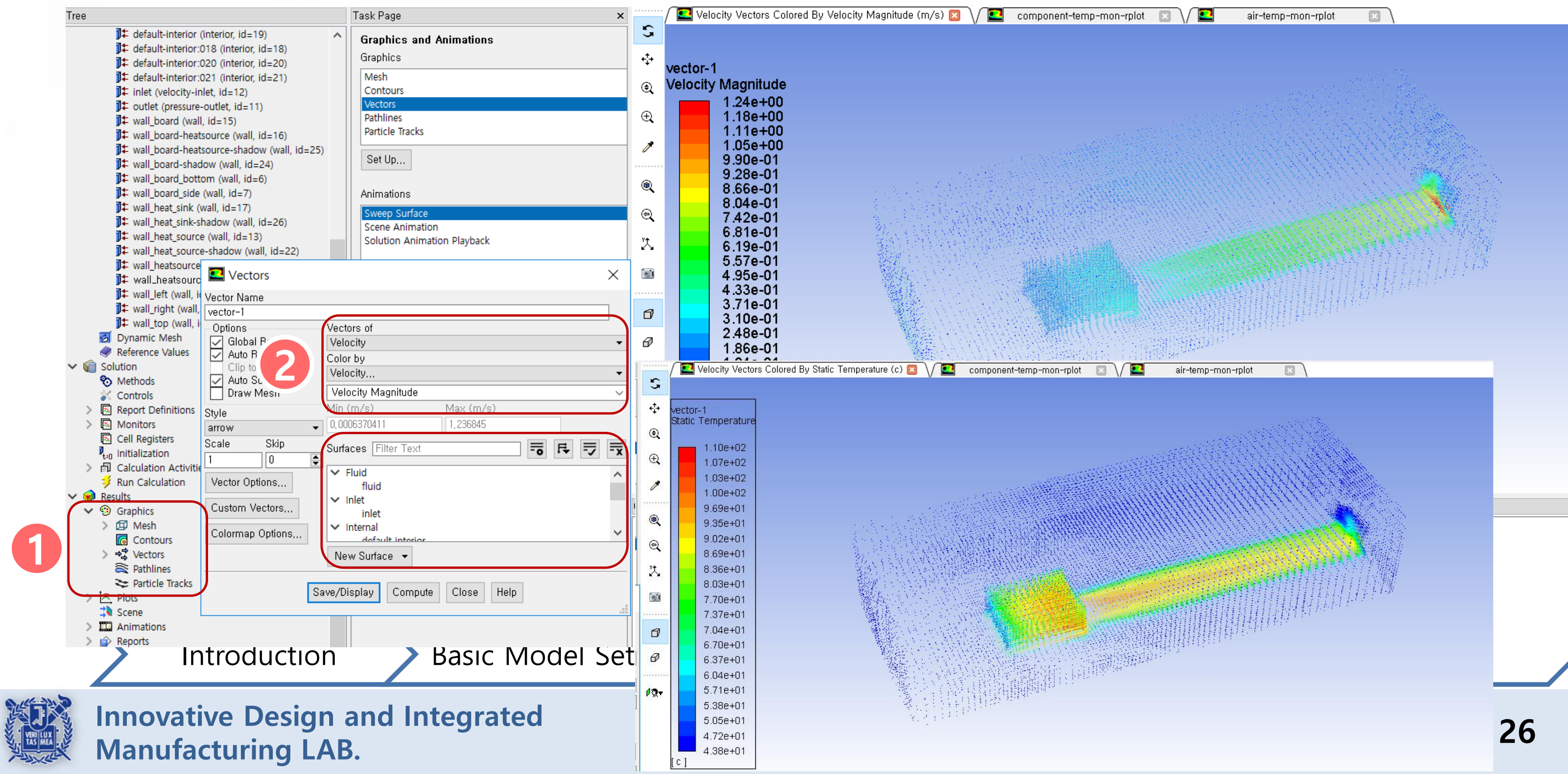
- Quick Post-processing
  1. Check total heat, mass balances
     - Reports → Fluxes → Set Up
     - Select the Inlet and Outlet, click compute
     - Net Results will be displayed
     - Check the value if it is low(it should be low)
  2. Total Heat Transfer Rate → choose all wall, Inlet, Outlet and then compute
  3. Check the difference with Energy Source(75W)
     - Difference between 75W and that one should be lower than<5







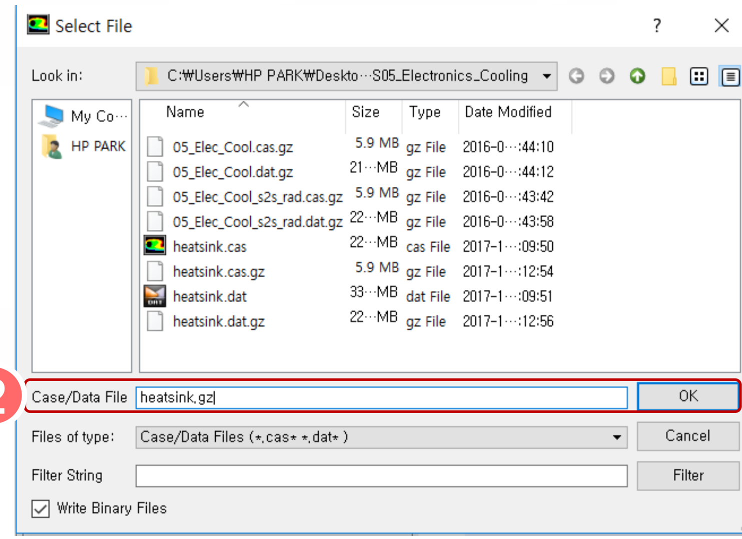
- Quick Flow Visualization
  1. Graphics → Vectors → Velocity(or Temperature)
  2. Select all walls → Save/Display



- Save Case and Data Files for Later use
    1. File → Export → Case & Data
- Work folder change and file name

2. If the file is saved by adding ".gz",
It can zip the files.

2



2

Summary

- From this example,
  - Conduction on solid is simulated.
  - Fluid and Solid conjugate heat transfer is considered.
  - Natural convection effect is simulated.

- Calculating the max. temperature is the main issue.
  - (Radiation effect can't be neglected. but, we skip the radiation process.)

- Before starting the analysis, we should clarify selection of the Mesh scale, Numerical method etc.
  - It can affect on Convergence speed, Stability.

THE END