Ansys Fluent Heat Transfer

Workshop: Flow Through a Louvered Fin Heat Exchanger

Release 2021 R2

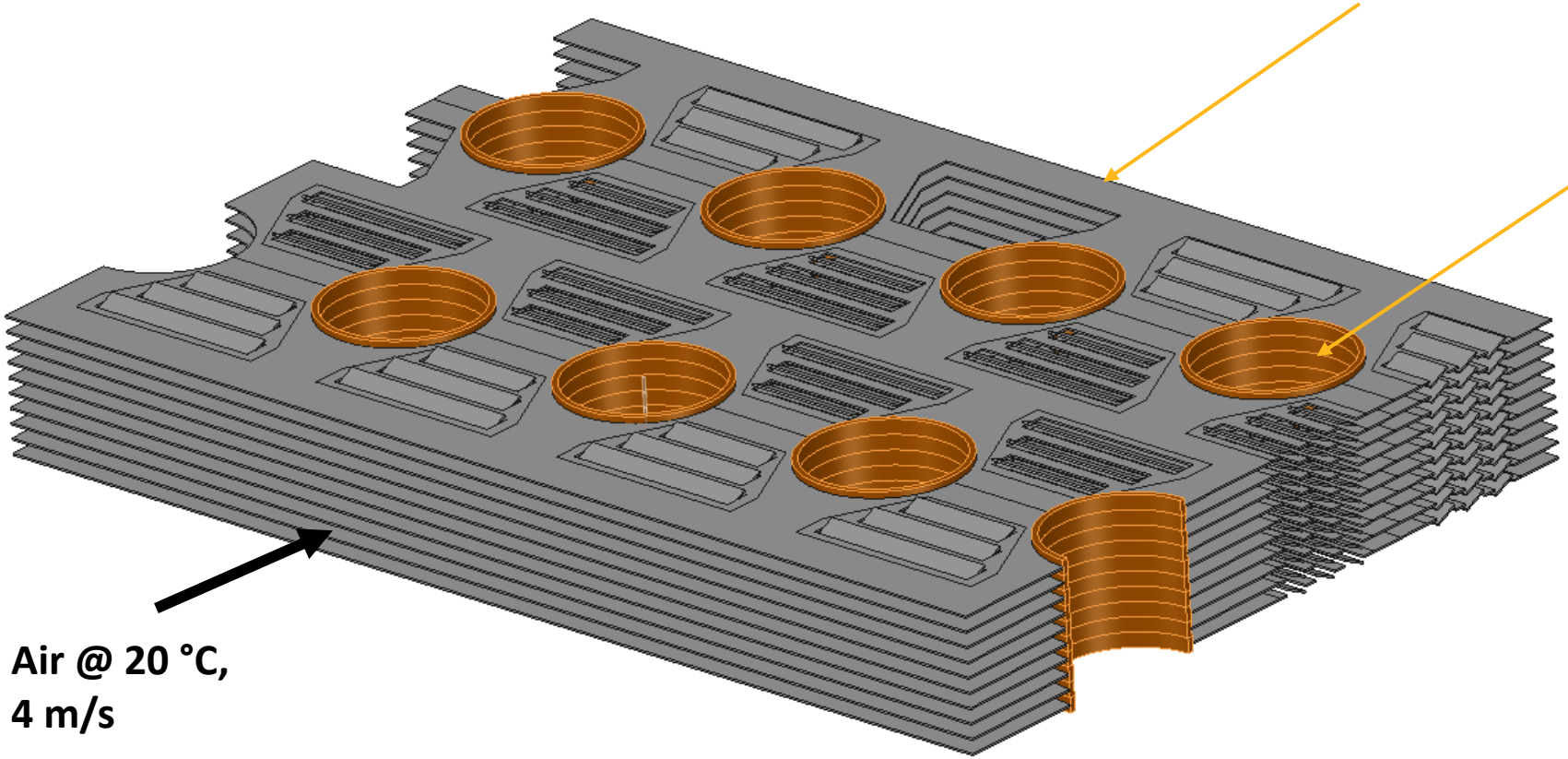
Introduction

- This tutorial demonstrates how to model forced convection in a louvered fin heat exchanger
- This tutorial demonstrates how to perform the following tasks:
  - Calculate the fin heat transfer rate
  - Use periodic boundaries to reduce the size of the computational domain
  - Use a convective thermal boundary condition to represent heat transfer inside the tubes
    - You solve only for flow and heat transfer on the air side
  - Examining the temperature and flow fields to understand the relationship between flow and temperature in the system

Prerequisites

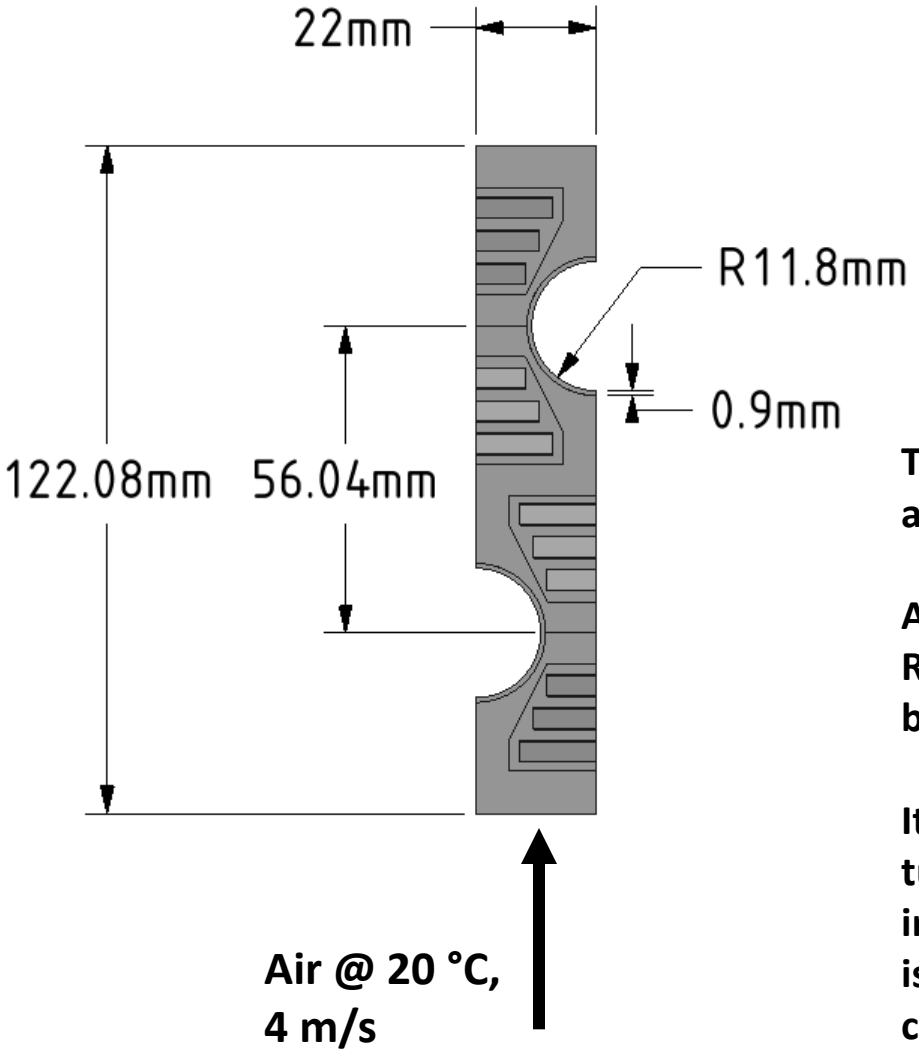
- This tutorial assumes that you are familiar with the Ansys Fluent interface and that you have a good understanding of basic setup and solution procedures.
- Note that some steps will not be shown explicitly.

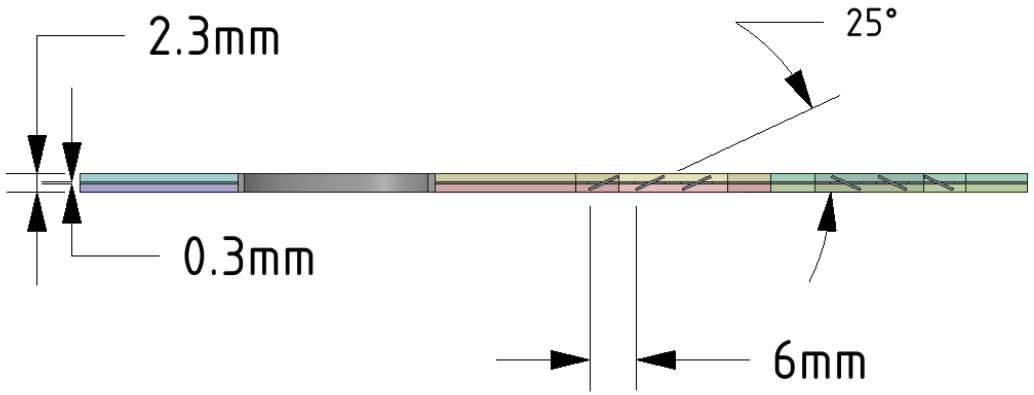
Problem Description: Geometry



The image is a section of a louvered fin heat exchanger. The flow of the air through a single fin passage will be simulated to determine the heat transfer and pressure drop characteristics. The model will make use of periodic and symmetry boundary conditions to account for the repeating fins and tube layout.

Problem Description: Dimensions





The tubes are arranged in a staggered layout with a spanwise pitch of 22 mm and a streamwise pitch of 56 mm. The fins are 3mm thick and have a pitch of 2mm.

Air at 20 °C approaches the heat exchanger with a velocity of 4.0 m/s. The Reynolds number based on the fin pitch is 1,470, so the air flow is considered to be laminar.

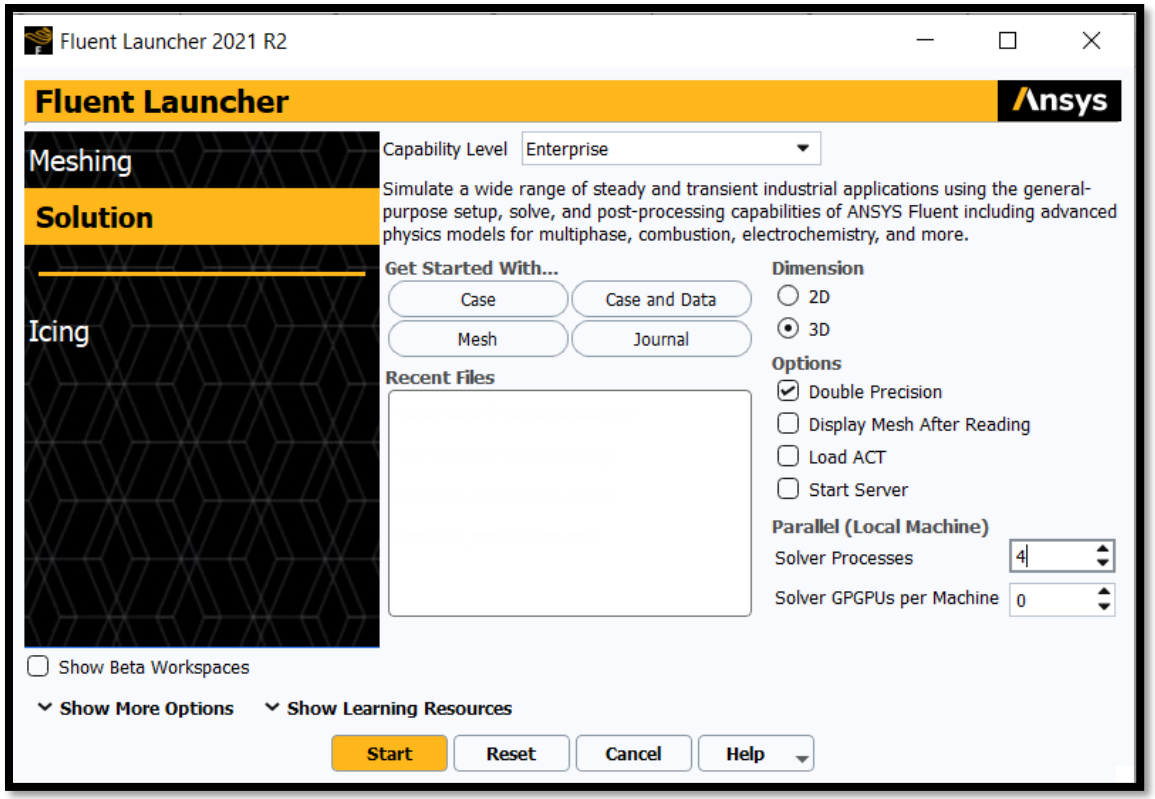
It is assumed that a 50% ethylene glycol solution at 80 °C flows inside the copper tubes at a velocity corresponding to a Reynolds number of 8,000. The liquid inside the tubes is not simulated. Instead a convective heat transfer coefficient is applied on the inner wall of the tube. The calculation of the heat transfer coefficient can be seen in the appendix.

Reynolds Number Calculation

- For estimation of the Reynolds number, we will use the fin spacing to determine the hydraulic diameter, d_h
  - For a planar channel, d_h = 2 x channel height
  - From slide 5, fins are 2mm apart, so d_h = 4mm
- Air properties
  - Density = 1.2 kg/m³, viscosity = 1.5e-5 kg/m·s
- Velocity between fins = 4.0 m/s x (2mm + 0.3 mm)/(2mm) = 4.6 m/s
  - HX frontal area includes fin thickness, velocity between fins is higher than approach velocity due to reduction in cross-sectional from fin blockage
- Reynolds number = 4.6 m/s x 1.2 kg/m³ x .004 m / 1.5e-5 kg/m·s = 1470
  - Internal flow, Re < 2,300 → laminar

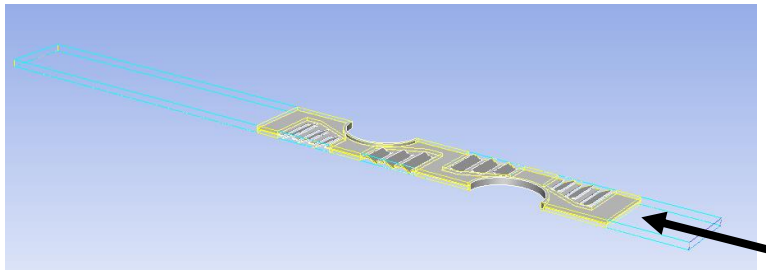
Preparation

- Copy the file hx-fin-2mm.msh.gz to your working directory
- Start the 3D double-precision version of Ansys Fluent
- If available, use 4 or more processors

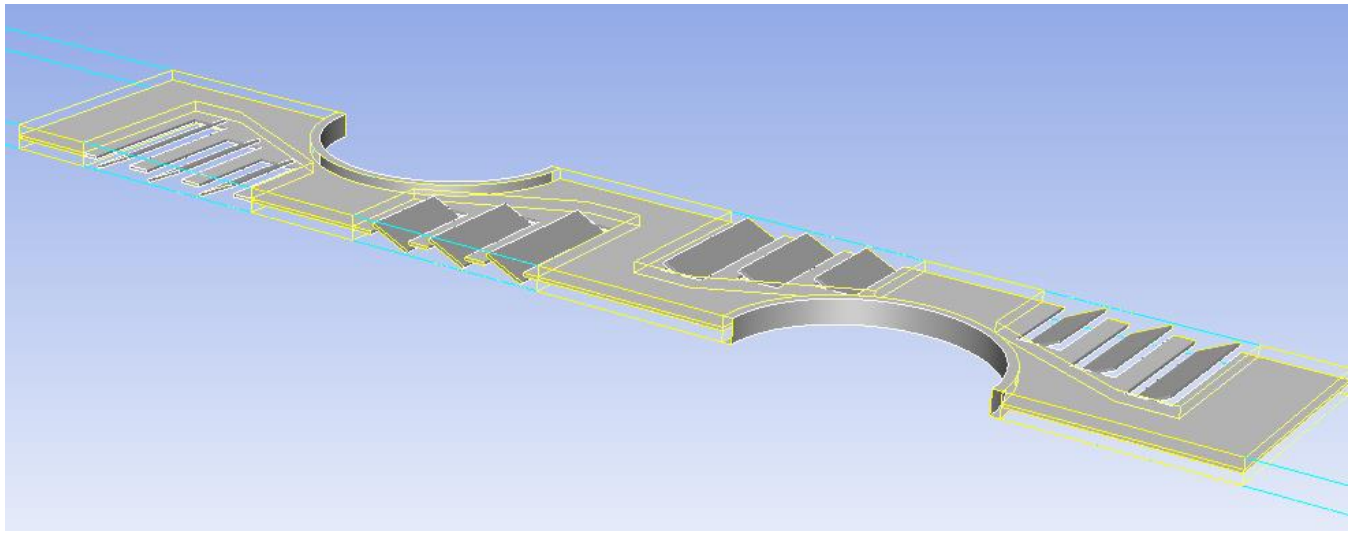


Setup and Solution

- Step 1: Mesh
  - Read the mesh file hx-fin-2mm.msh.gz

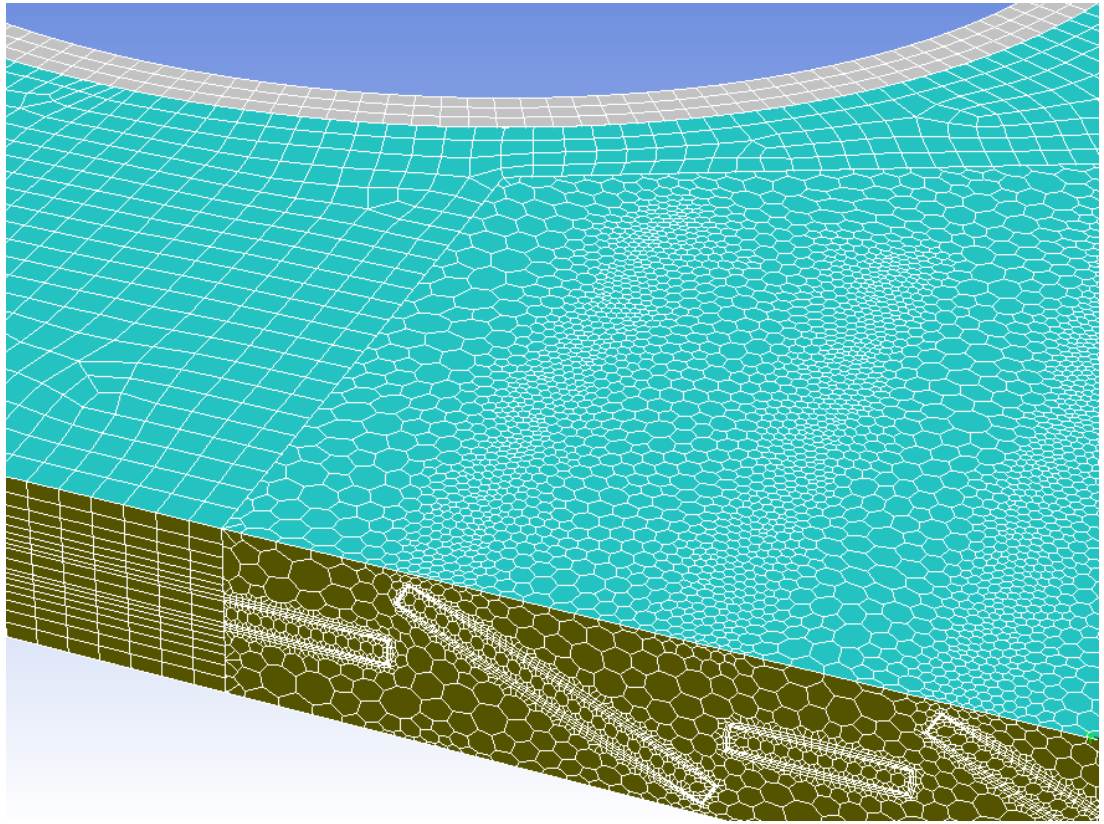


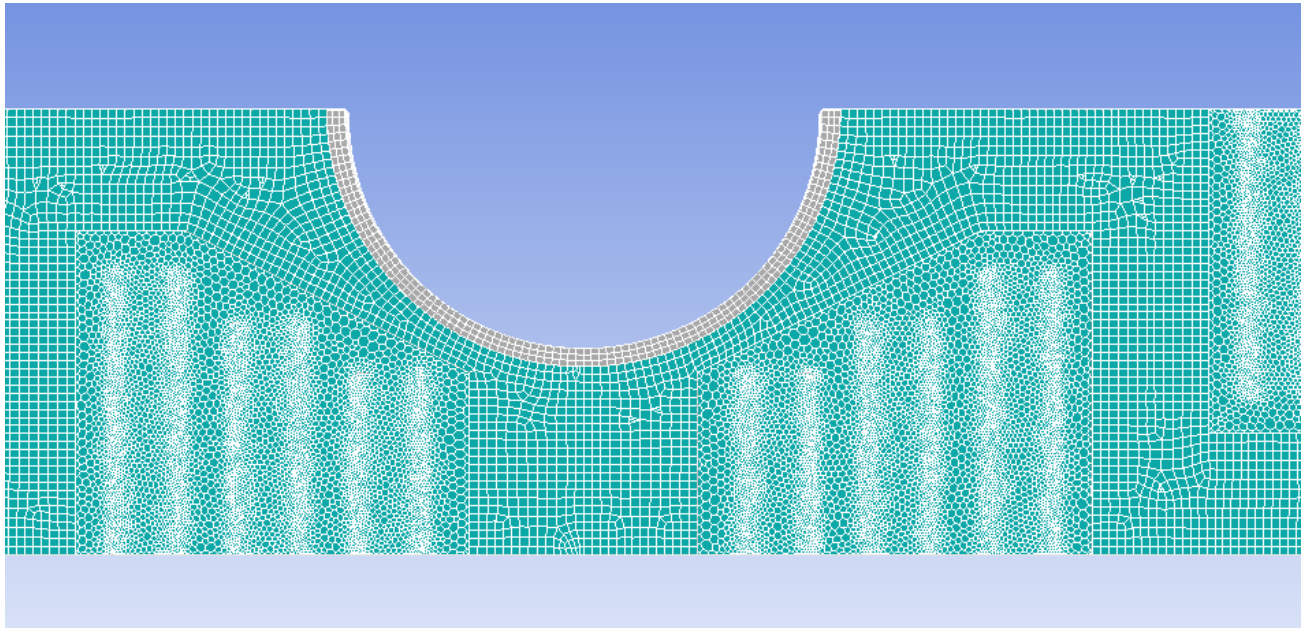
Flow Direction



The domain comprises a single fin passage in a louvered fin heat exchanger. Louvers are used to disturb the boundary layer and enhance heat transfer compared to a flat fin. A heat exchanger might have hundreds of such fins, which is not computationally tractable. Therefore, periodic boundaries (which appear blue in the picture) are used to reduce the domain to a single fin passage. Flow exiting through a periodic boundary re-enters the domain at the corresponding location on the opposite side, with the same velocity, temperature, pressure, etc. In this model, each periodic boundary is located midway between the surfaces of two neighboring fins.

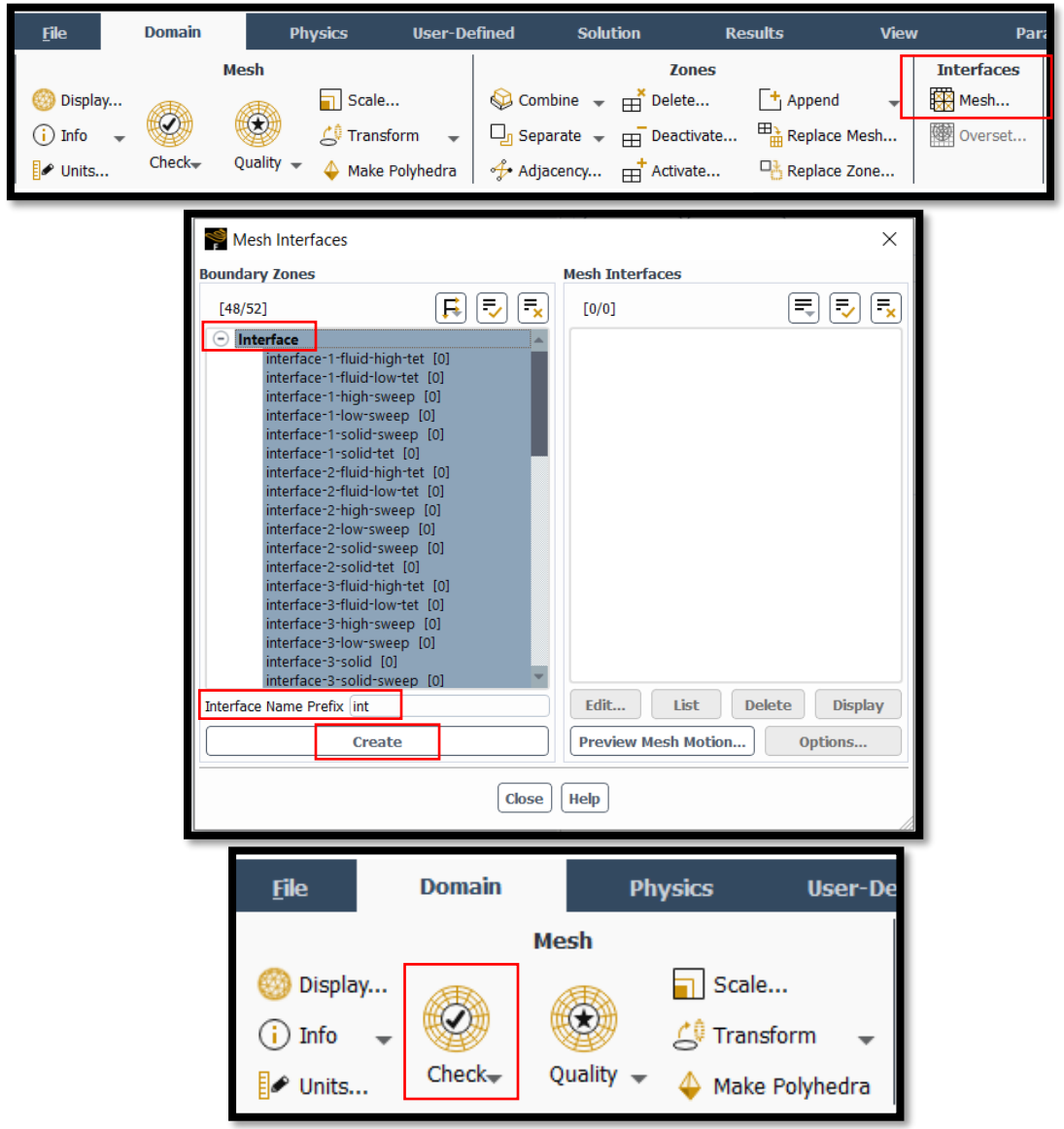
Display Mesh





Several non-conformal interfaces are used in order to mesh as much of the domain as possible with hexahedral cells. The reason for doing so is to limit the cell count. Regions around the louvers, which cannot be easily decomposed for hex meshing, are meshed with polyhedral cells.

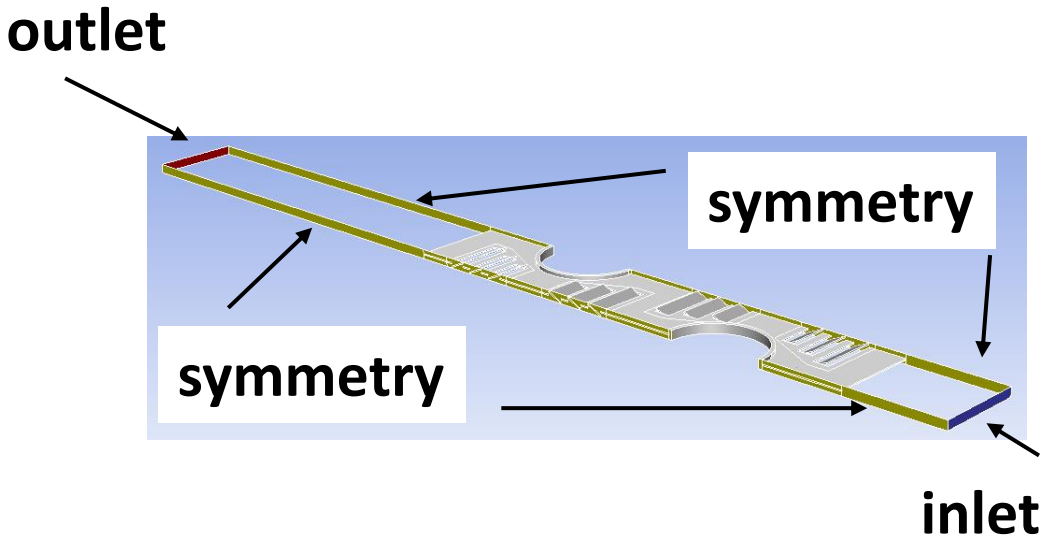
Mesh Interfaces



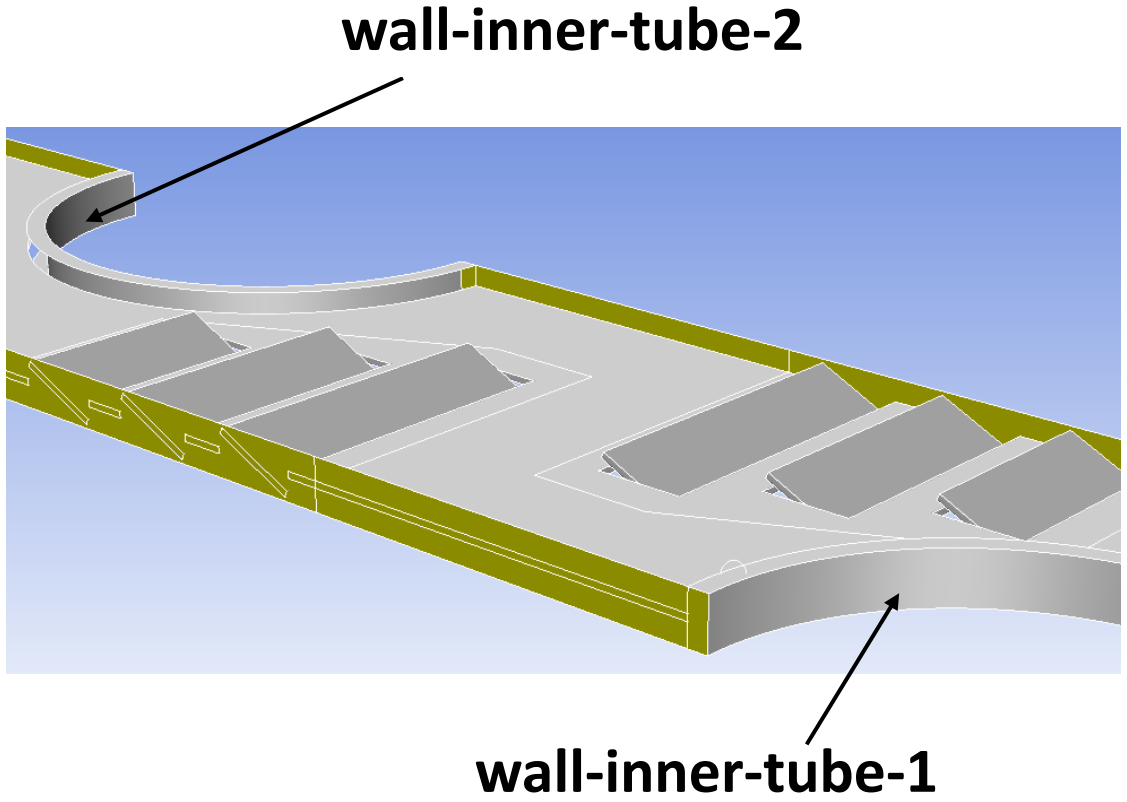
- In the Setting Up Domain tab, under Interfaces, select Mesh…
- In the Mesh Interfaces panel, select all unassigned interface zones, enter a prefix name (here "int") and click Create
  - Auto creation of mesh interfaces was first introduced in R18.1.
  - Perform a mesh check
  - This is imperative when using mesh interfaces and/or periodic boundaries, both of which apply here
  - As long as you do not see any error or warning messages, it is fine
    - If there was a negative volume (or any other error) in the mesh it would throw an impossible to miss error message

Visually inspect the boundaries

- Take a moment to identify the various boundaries



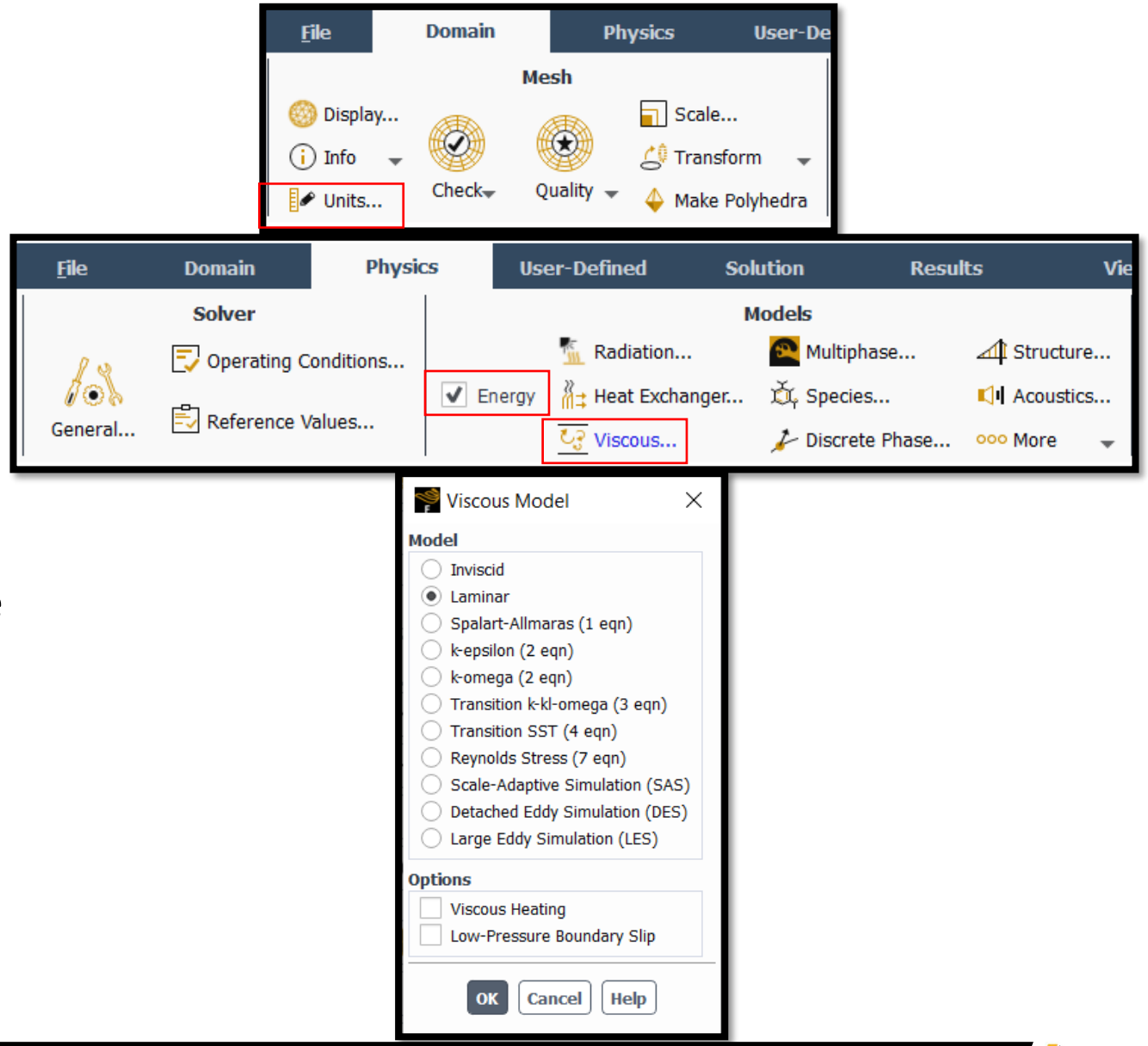
Note: Periodic boundaries are not displayed here so that interior of domain is visible. All gold faces on either side of the domain are symmetry boundaries.



Note: Other wall zones (e.g. those forming the fin) are not explicitly labeled on the picture. The names of these zones begin with "wall-fluid.." and they are all coupled walls (e.g. wall + wall-shadow)

Units and Setting Up Physics (Energy and Viscous)

- In Units… set the temperature units to be Celsius
- In Physics, activate ***Energy***
- In Viscous Model panel, select ***Laminar***
  - As shown previously, based on the fin spacing, the Reynolds number is 1470, so the air flow will be calculated as laminar



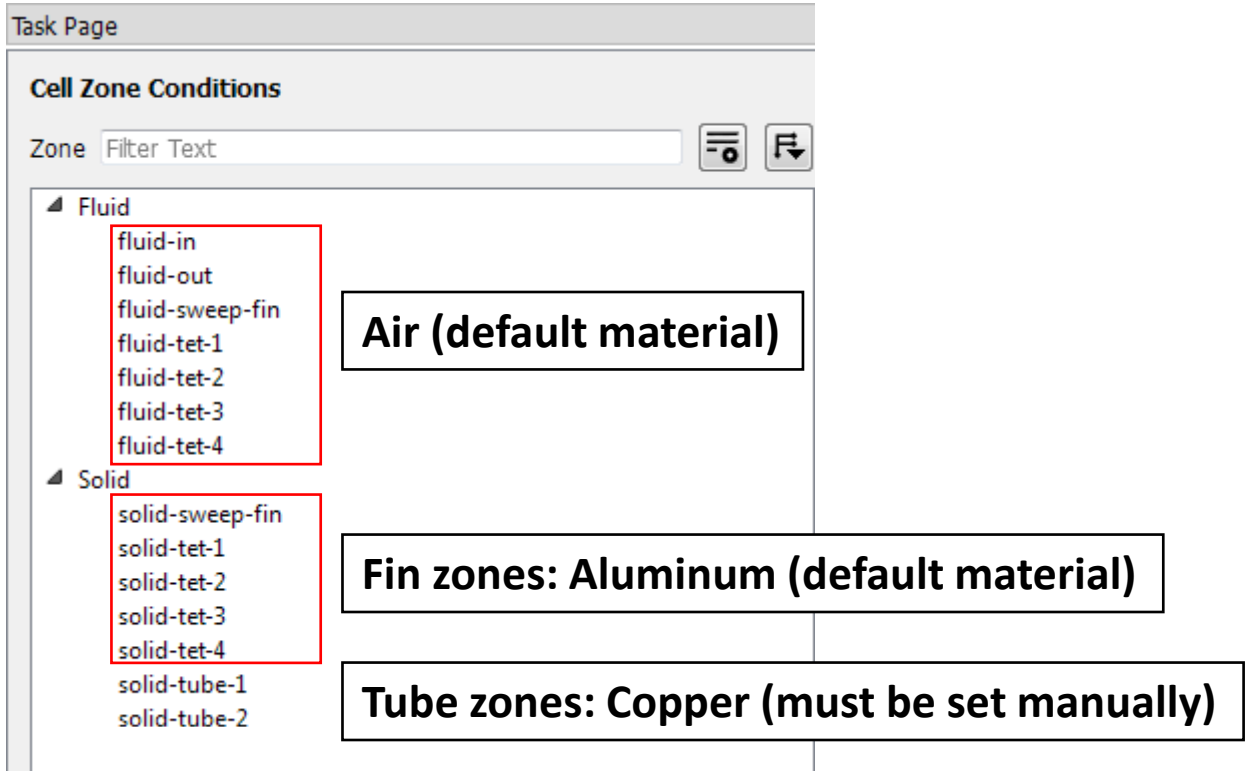
Setting Up Physics: Materials

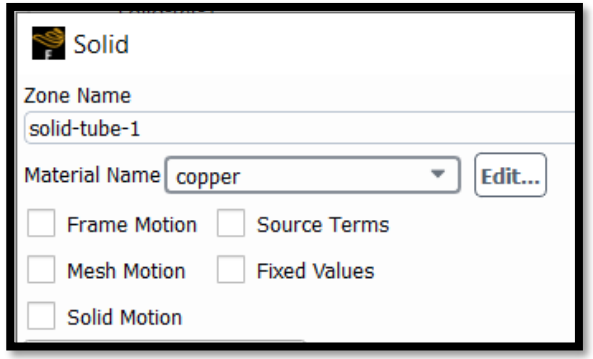
- In the Materials Panel, copy Copper from the Fluent Database
- Set properties for air, copper and aluminum as follows:

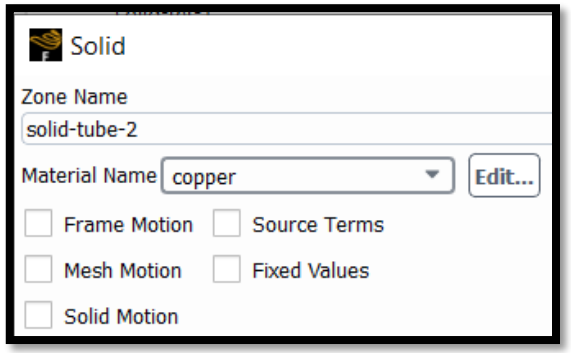
	Air	Aluminum	Copper
Density (kg/m^3)	1.2	2719	8978
Viscosity (kg/m·s)	1.50E-05	n/a	n/a
Thermal Conductivity (W/m·K)	0.026	200	340
Specific Heat (J/kg·K)	1006	871	381

Setting Up Physics: Cell Zone Conditions

- Air is the default material, so no input is required for the cell zone conditions for any fluid zone
- The fin is aluminum, which is the default material so no input is required for any of the 5 cell zones which comprise the fin
- The **tubes** are made of **copper**, therefore open the cell zone conditions panel for each tube and change the material to copper

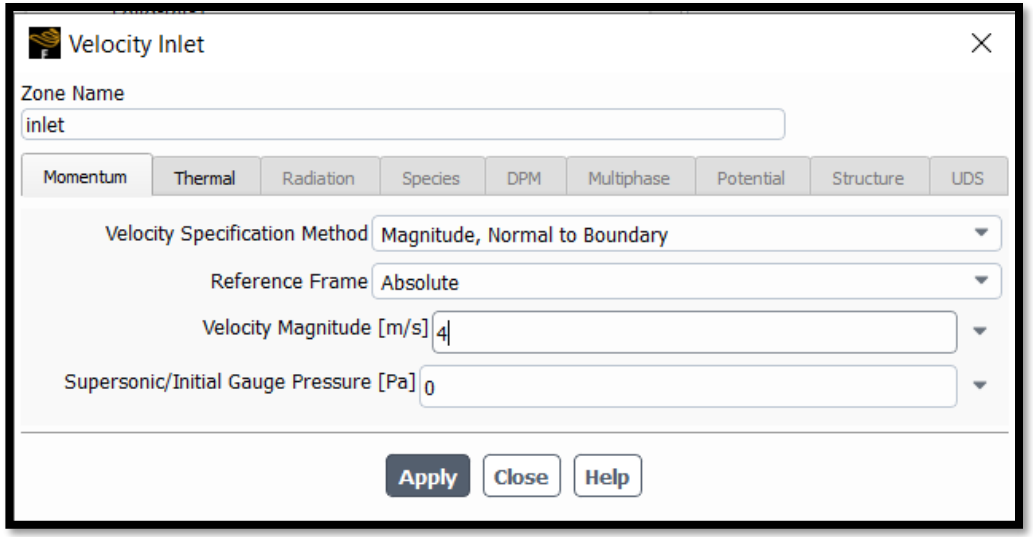


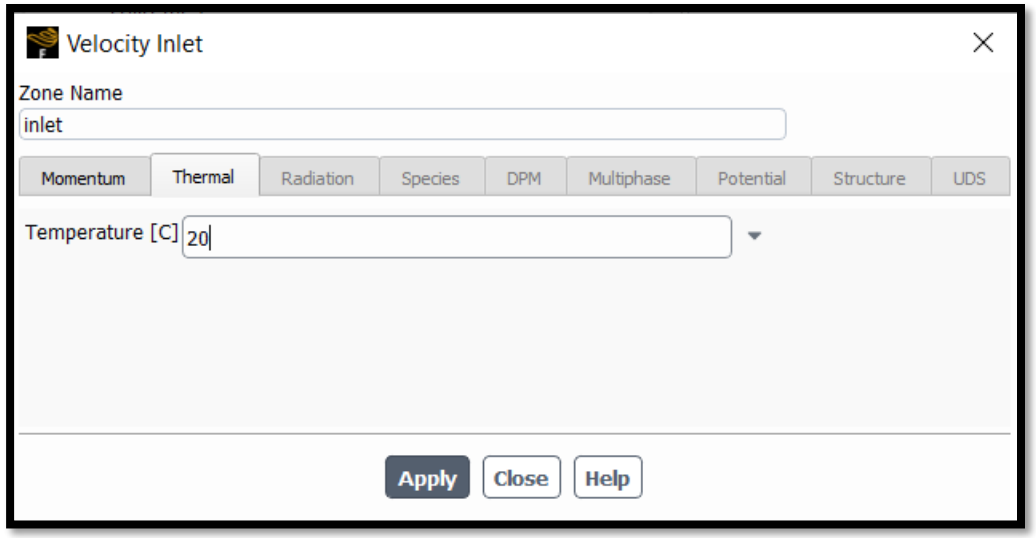




Setting Up Physics: Boundary Conditions for Inlet and Outlet

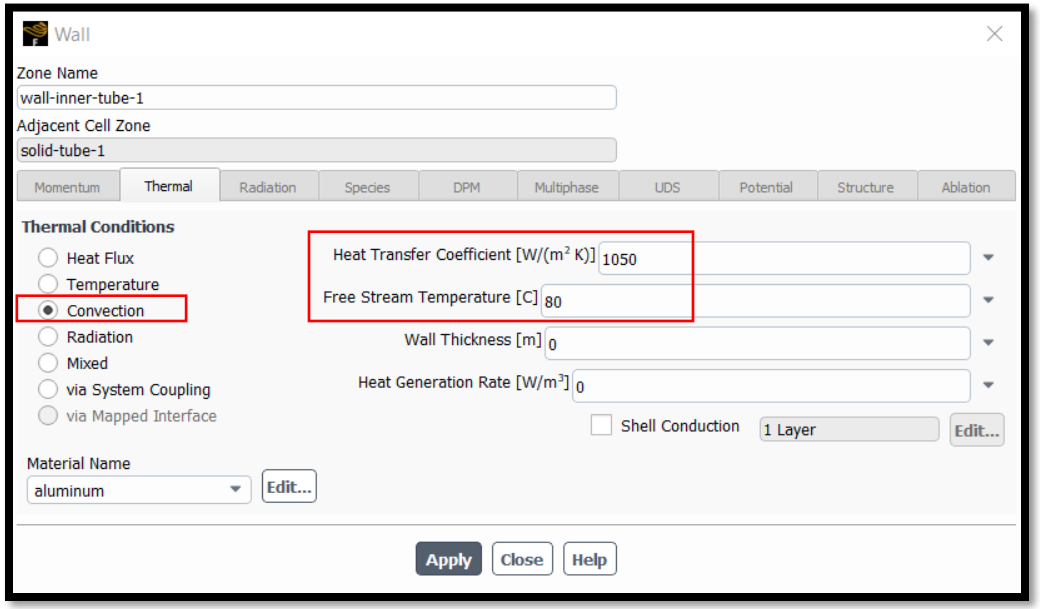
- At the inlet, set the velocity to 4.0 m/s and the temperature to 20 °C
- At the outlet, set the backflow temperature to 20 °C
  - Not shown

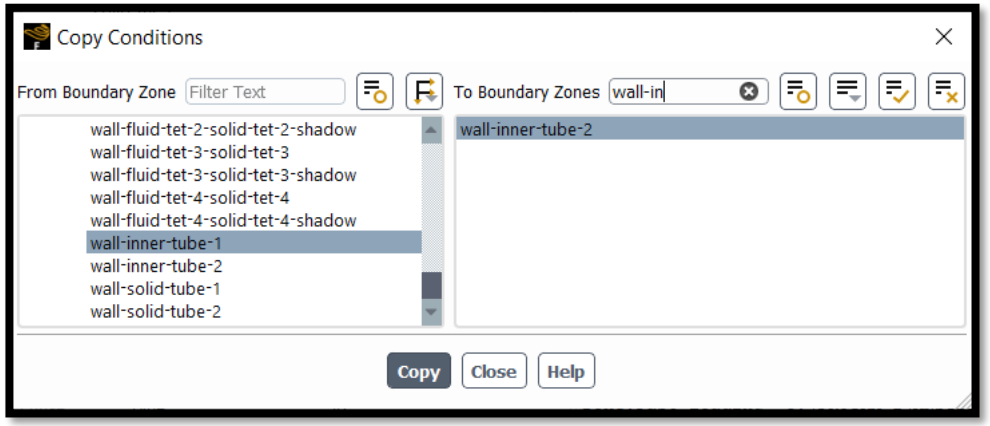




Setting Up Physics: Boundary Conditions for Tubes

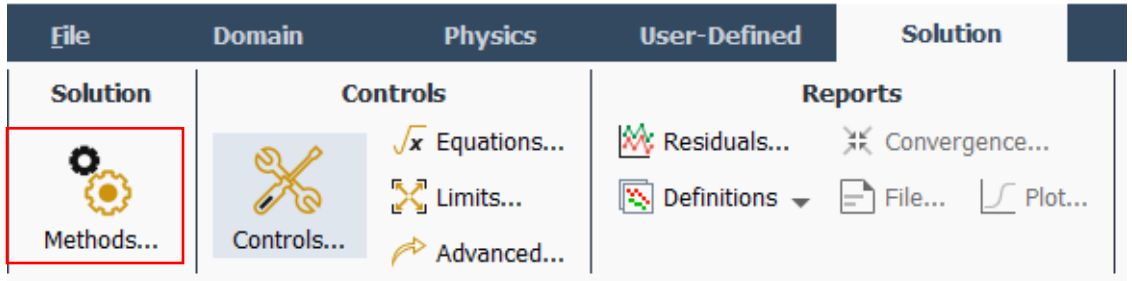
- Set the thermal boundary conditions for **wall-inner-tube-1** as shown
  - See appendix for calculation of heat transfer coefficient
- Copy the same conditions to **wall-solid-tube-2**
- Other wall boundaries will use default wall b.c. settings so it is only the two tube wall boundaries that need to be set
- Save the case file: *hx-fin-2mm.cas.h5*

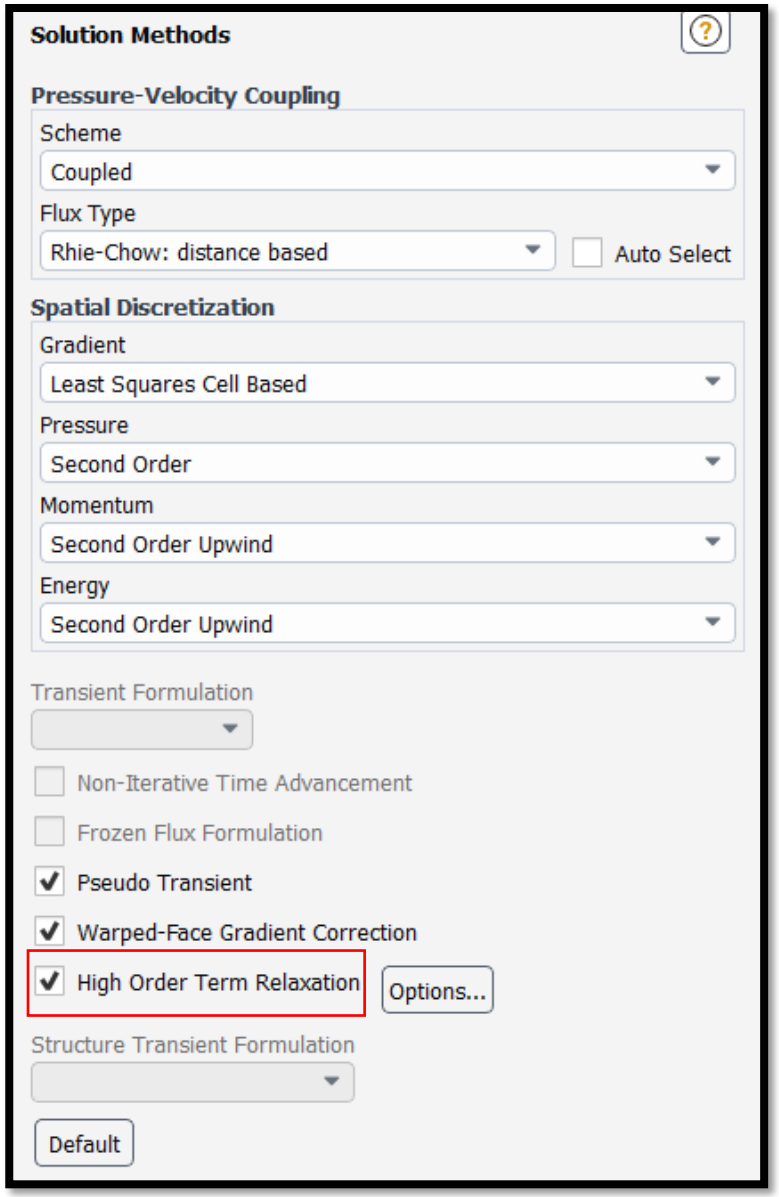




Solving: Methods

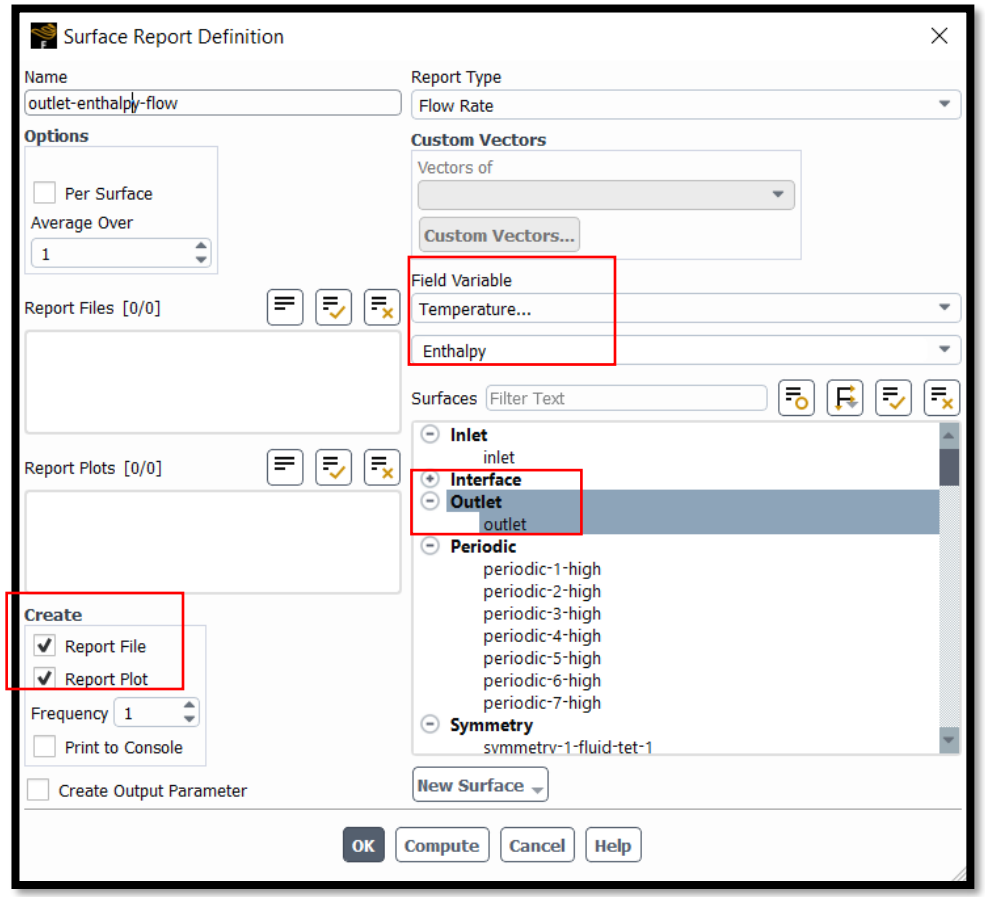
- In the Solution tab, click on Methods, keep the default settings with the addition of activating high order term relaxation options

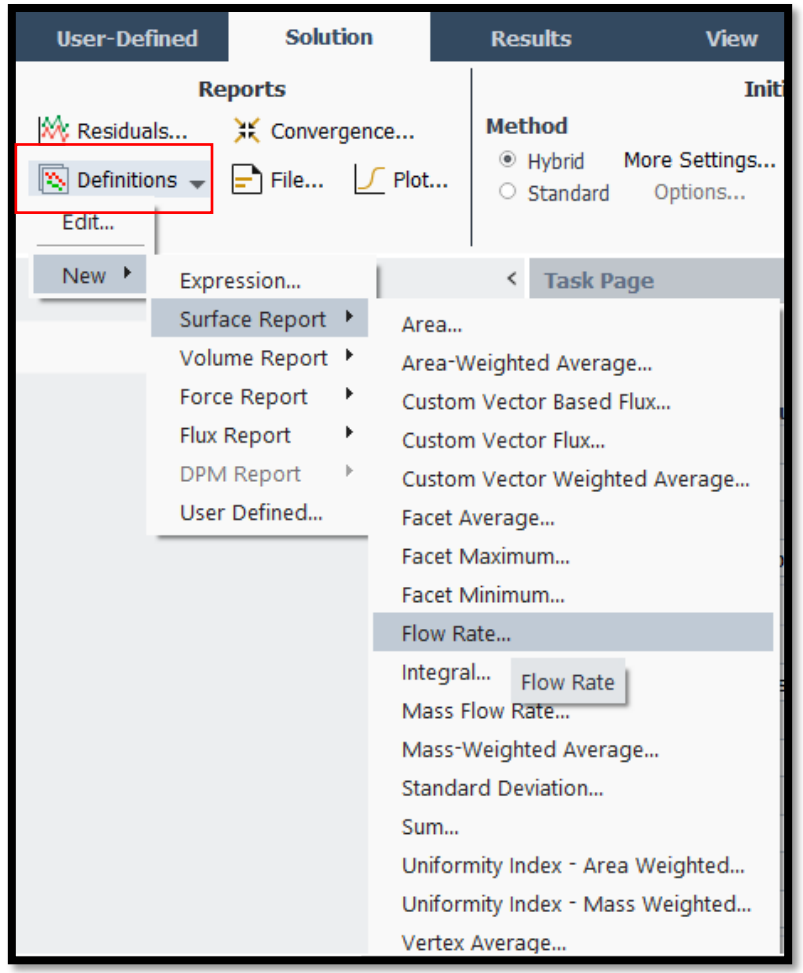




Solving: Report Definitions

- Create a Report Definition to plot the enthalpy flow rate at the outlet
  - Report Plot and Report File are selected

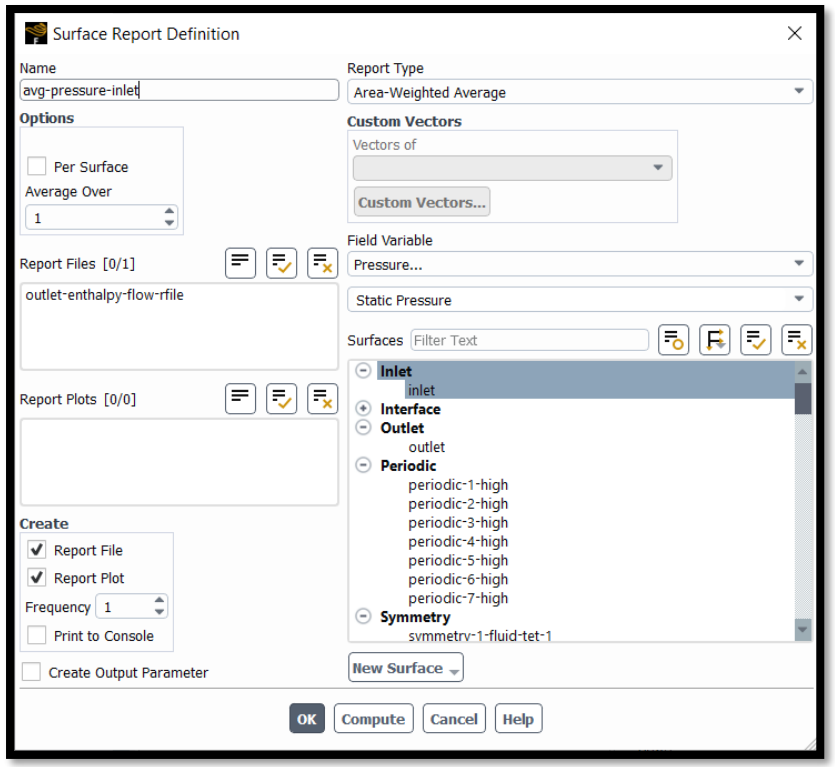


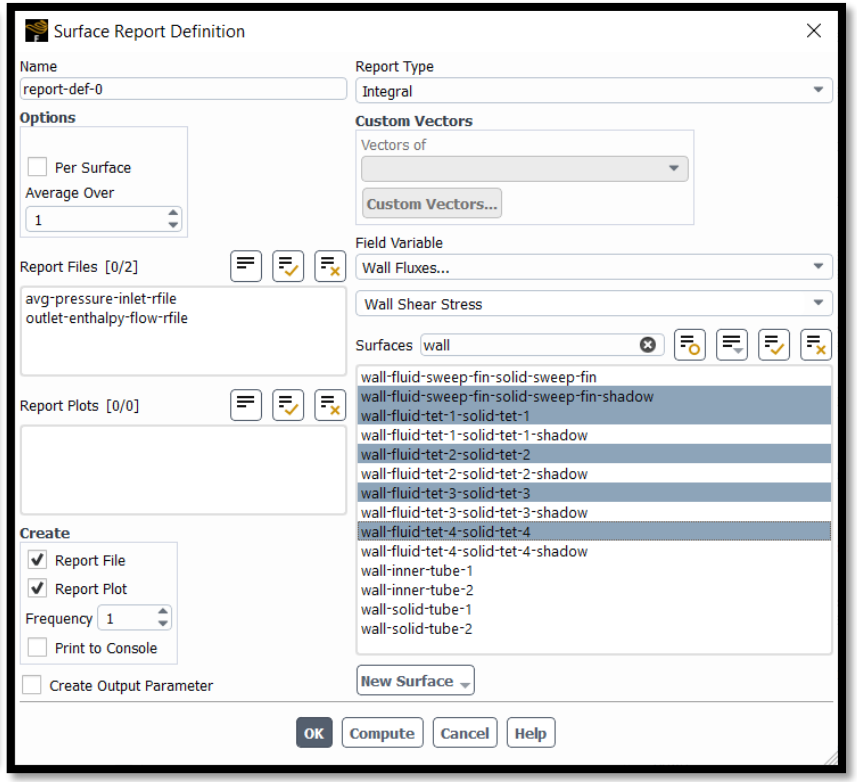


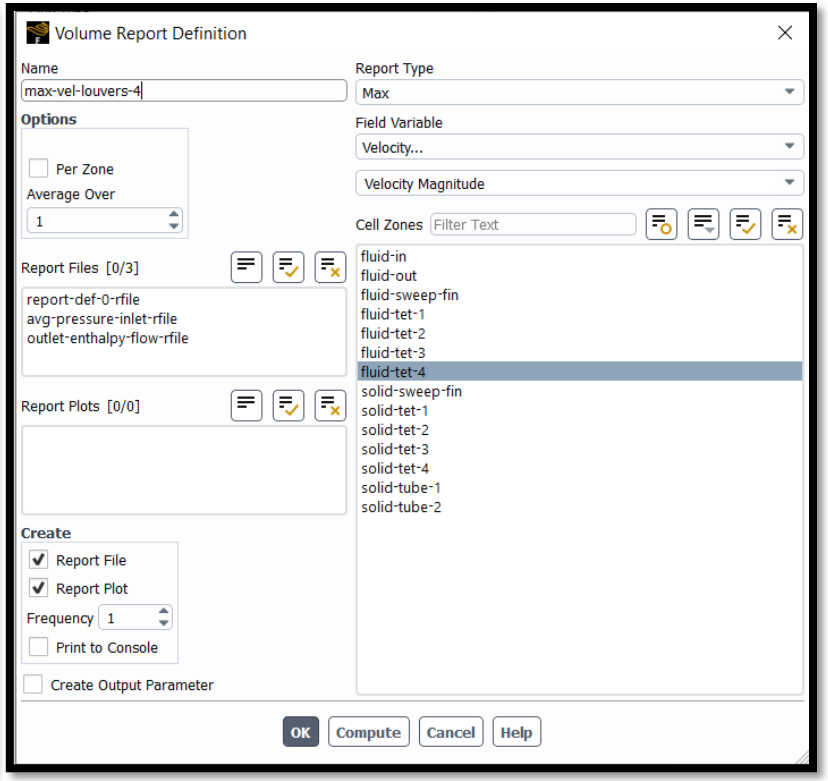
Plotting the enthalpy flow rate at the outlet is a way to check that the energy balance has stabilized

Solving: Report Definitions

- Create additional report definitions as shown







Hint: typing "wall" in the filter box (next to "Surfaces") will make selection easier

Solving: Initialization and Modify Zones

- Click Initialize
  - Default hybrid initialization method will be used
- Note the message that appears in the console

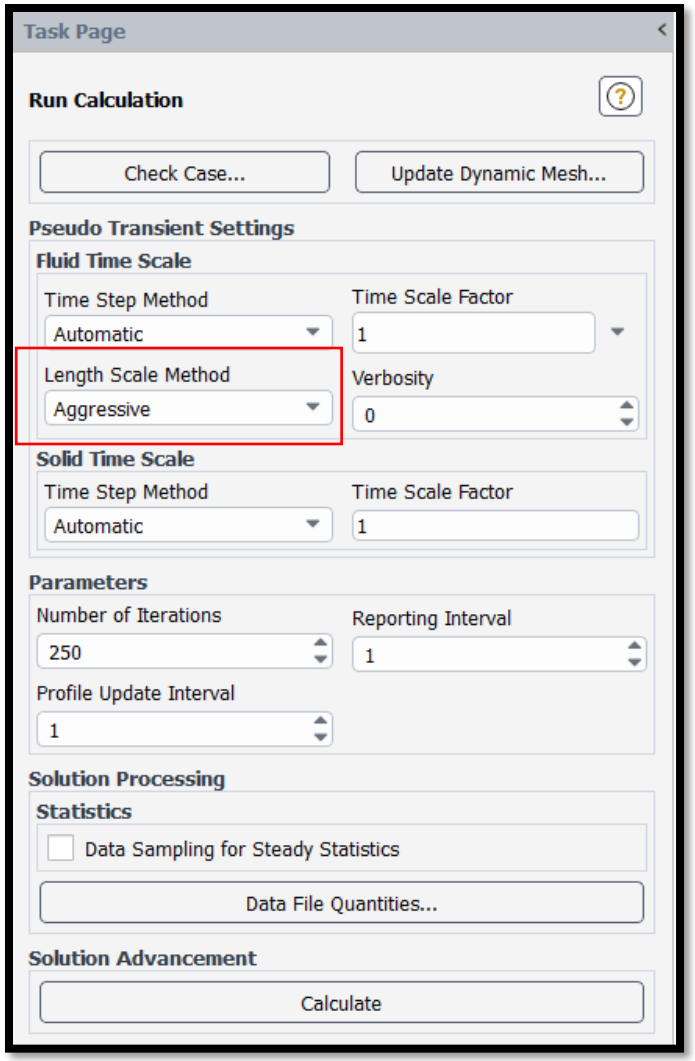
```
Warning: zone of type interior found between different solids!
  Material of cell zone 70 is aluminum, while material of cell zone 7 is copper.
Warning: zone of type interior found between different solids!
  Material of cell zone 70 is aluminum, while material of cell zone 8 is copper.
      This will adversely affect the solution.
      It is recommended that you fix this issue via the TUI command
      /mesh/modify-zones/slit-interior-between-diff-solids
Initialize using the hybrid initialization method.

Checking case topology...
-This case has both inlets & outlets
-Pressure information is not available at the boundaries.
 Case will be initialized with constant pressure
```

- Execute the TUI command suggested by the message, save the case file and initialize again
  - This command creates a wall/wall-shadow boundary between the tubes and the fin, which allows a more accurate calculation of conduction between different materials

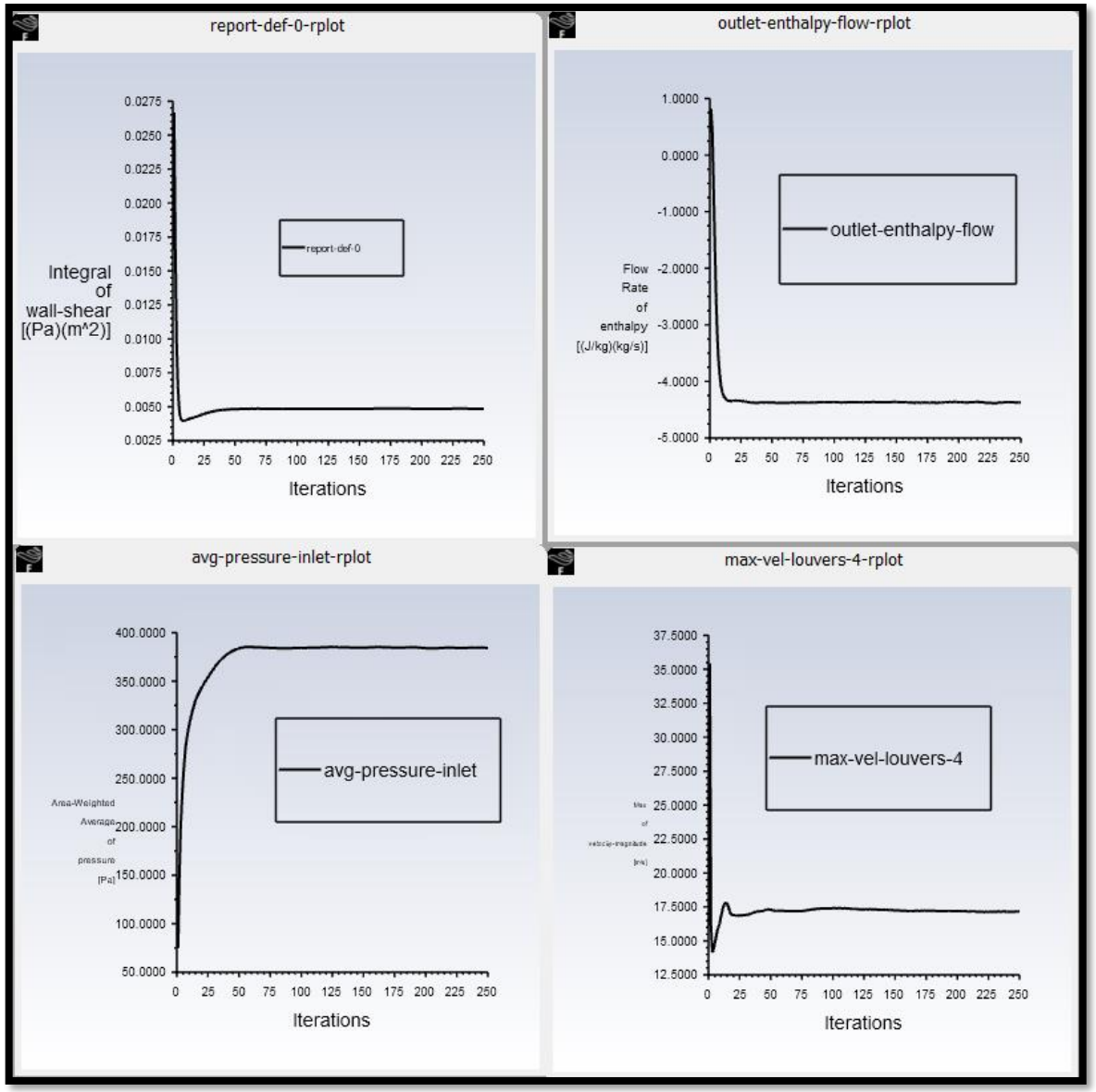
```
> /mesh/modify-zones/slit-interior-between-diff-solids

  convert interior zone 150 to wall.
  convert interior zone 122 to wall.
Note: Slitting sliding interface zone 150 into a coupled wall with shadow zone 376.

Note: Slitting sliding interface zone 122 into a coupled wall with shadow zone 377.

        creating int:21:interface-tube-1-solid-a.::interface-tube-1-solid-a-shadow
        creating int:24:interface-tube-2-solid-a.::interface-tube-2-solid-a-shadow
```

Solving: Run Calculation

- Click Initialize
  - Default hybrid initialization method will be used
- In the Run Calculation area, click Advanced and then choose Aggressive for the Length Scale Method
- Enter 250 iterations and click Calculate

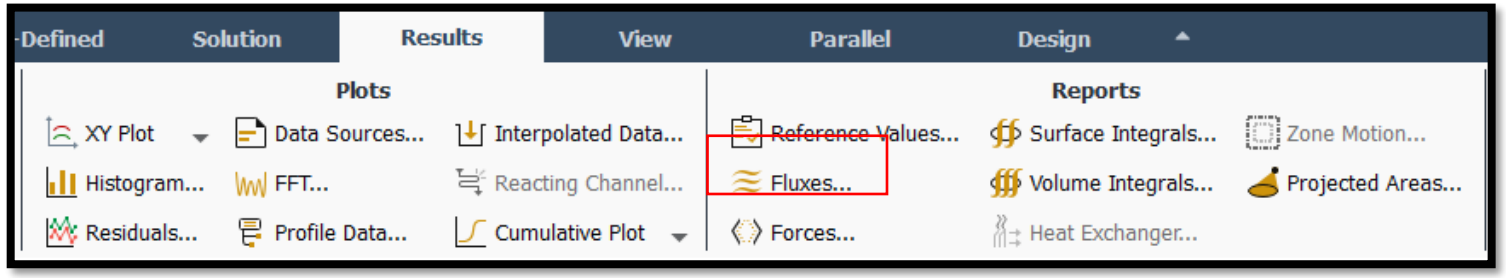


Solution Convergence Behavior

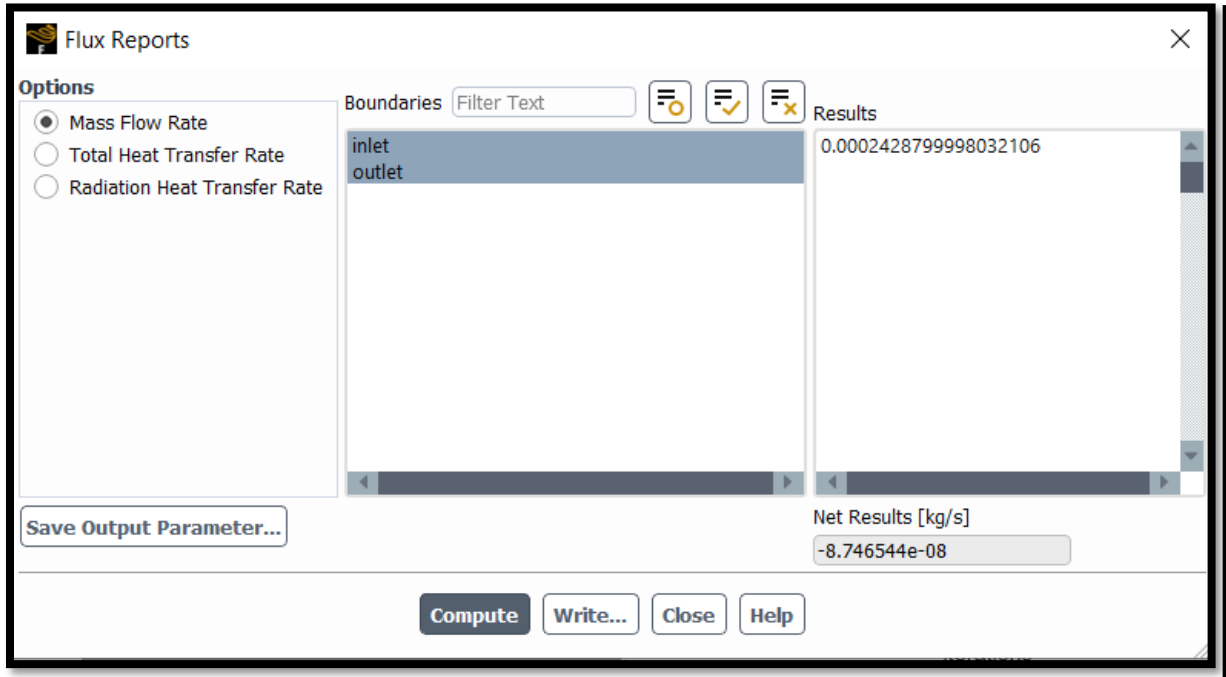


The report plots indicate the solution is no longer changing

Postprocessing: Mass and Energy Balances

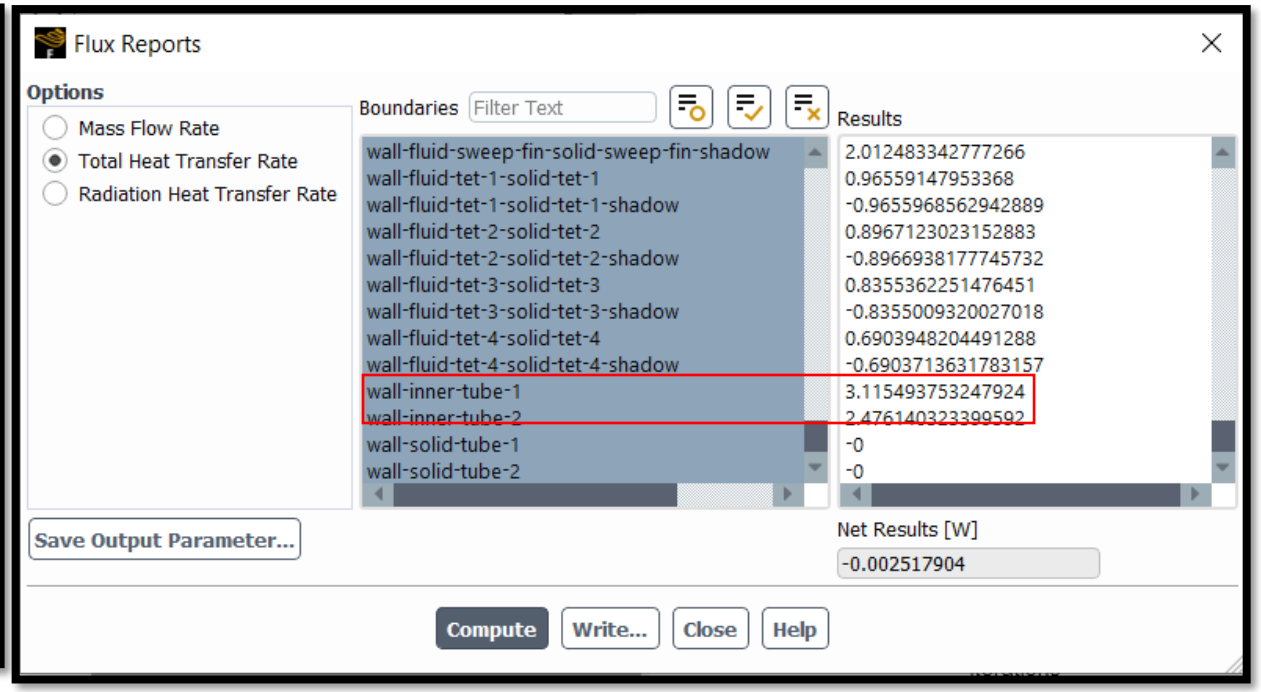


- Mass balance



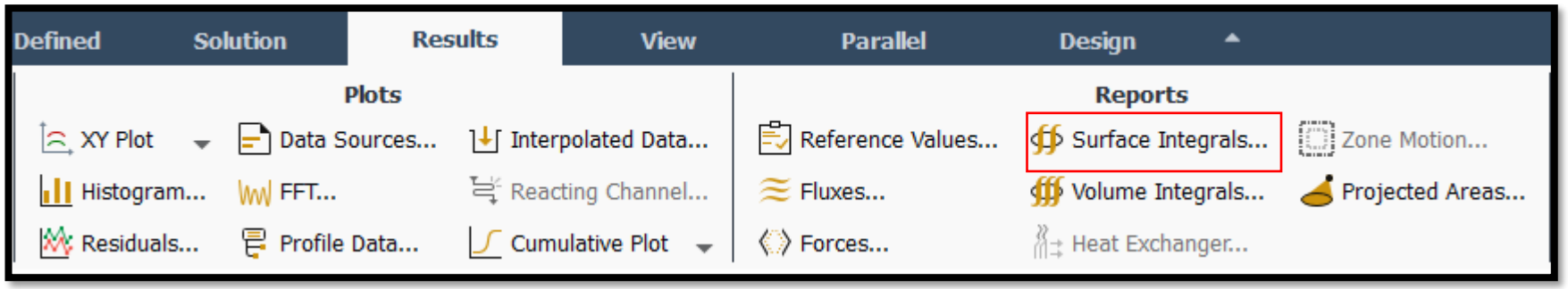
Mass imbalance is ≈ 0.04 % of inlet mass flow rate

- Energy balance

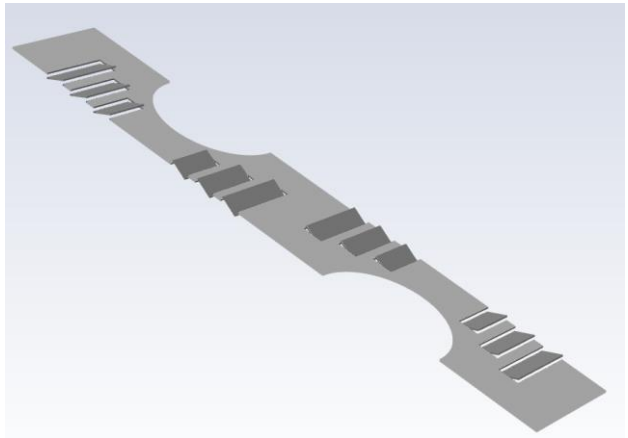


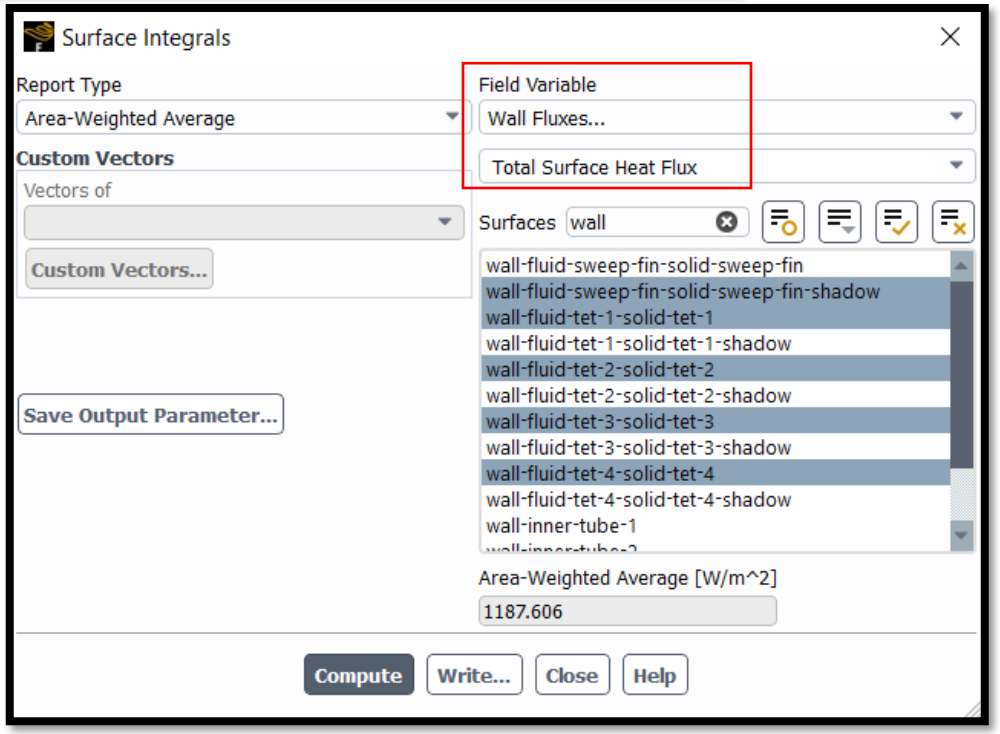
Heat transfer rate from tubes to air is 5.6 W
Energy imbalance is .04 % of this value

Postprocessing: Average Heat Flux



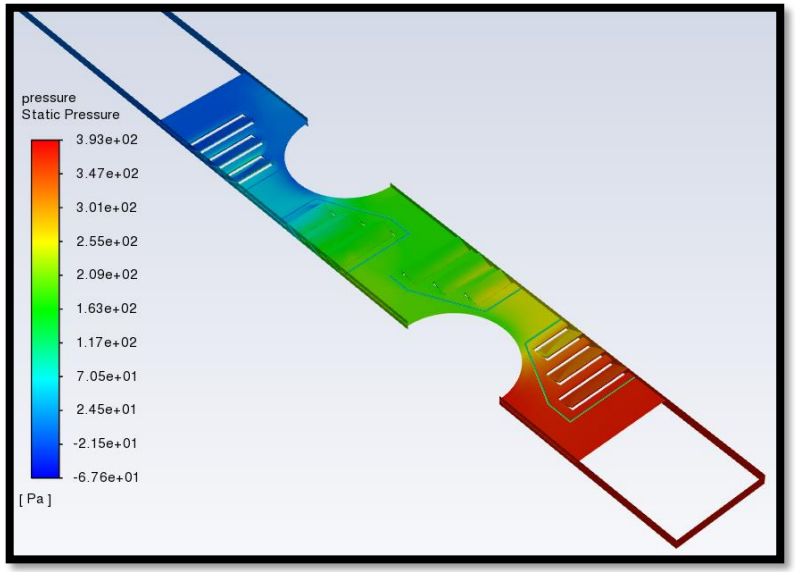
- In the surface integrals panel, select the fin surfaces (same as the report definition on slide 19) and compute the area weighted average heat flux





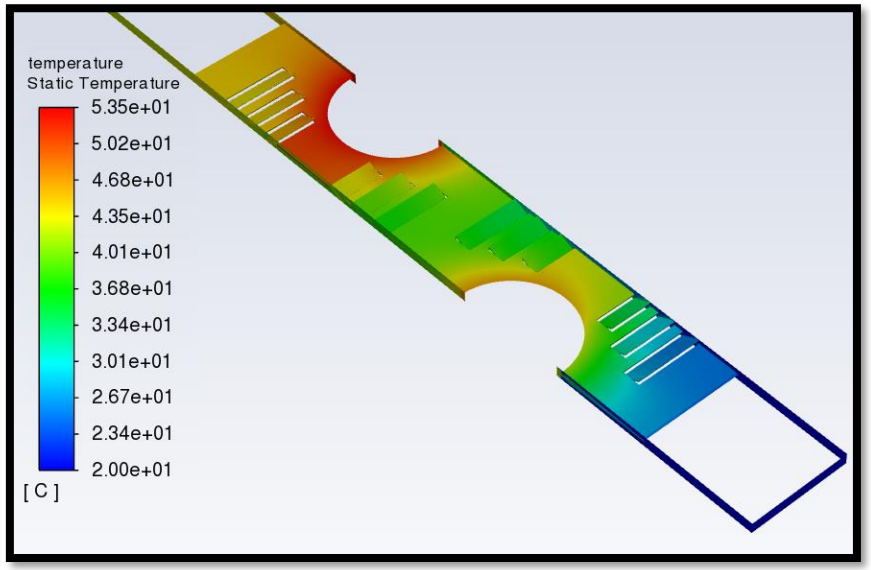
Average heat flux is ~ 1200 W/m²

Postprocessing: Pressure Contours on Fin

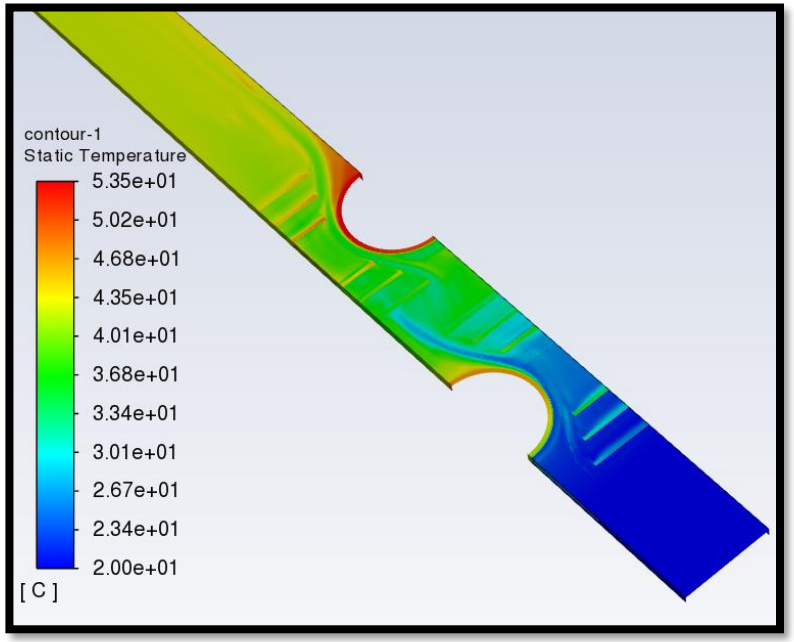


Contours of static pressure on fin surface.

The pressure drop across the fins is a quantity of interest for heat exchanger design. It would be possible to do additional simulations to determine the pressure drop as a function of inlet velocity

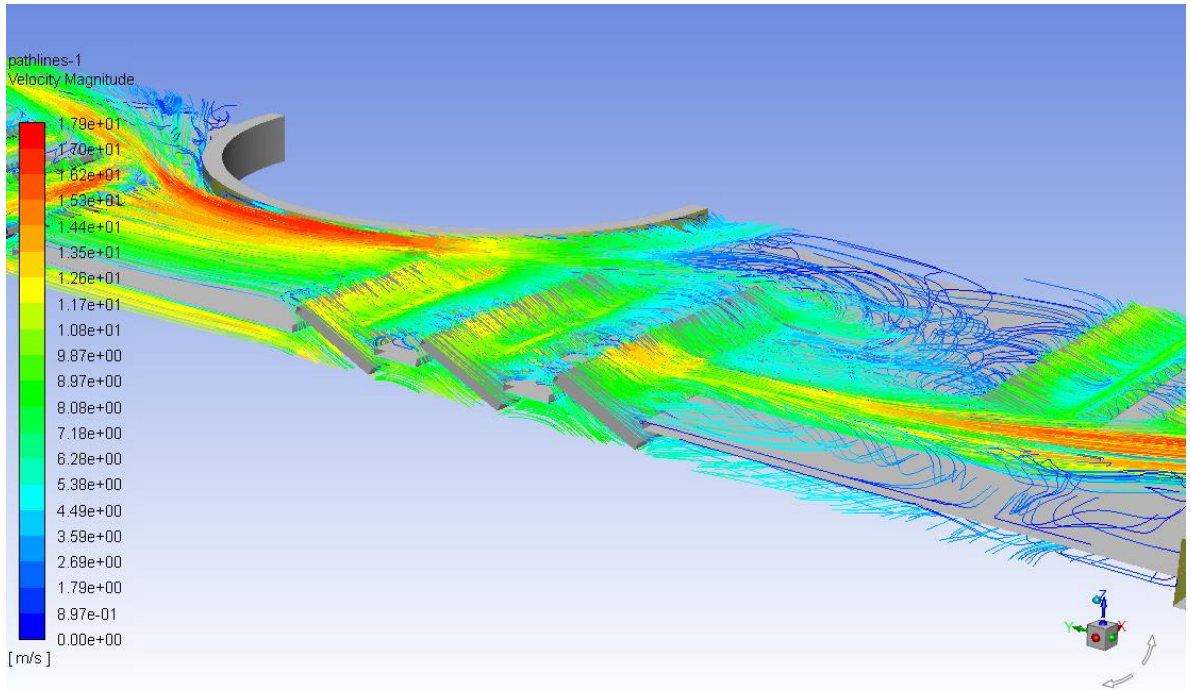


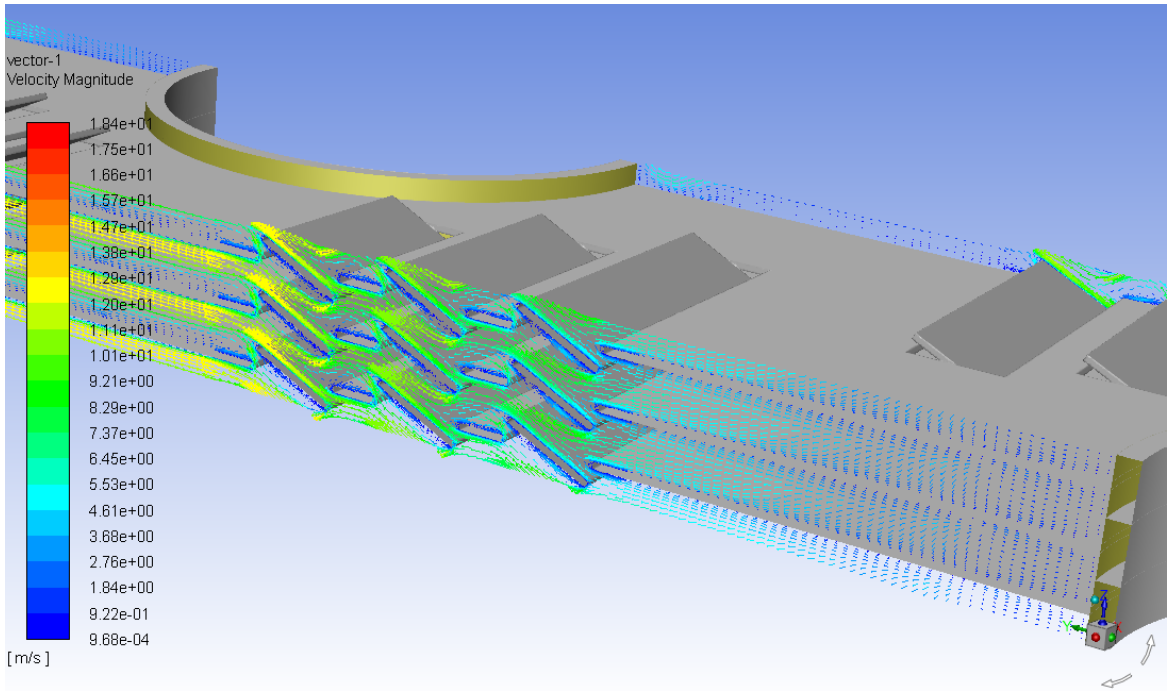
Contours of temperature on fin surface.



Contours of air temperature on a plane 0.5 mm above the fin surface. As expected, the highest air temperatures are observed in the wake of the downstream tube

Postprocessing: Flow Field





Summary

- In this workshop you
  - Set up and solved a problem with forced convection and conjugate heat transfer
  - Used a combination of periodic and symmetry boundaries to reduce the extents of the computational domain to a single flow passage
  - Performed an energy balance on the domain, comparing the heat transfer from the tube walls with the net imbalance in the flux reports panel
  - Used post-processing to determine pressure drop across the heat exchanger and average heat flux from the fin surface

Appendix

Calculation of heat transfer coefficient used in tube wall boundary conditions

- The Dittus-Boeltter correlation can be used to determine the heat transfer coefficient for flow in pipes

Properties for 50% Ethylene Glycol				
(source: http://www.mhtl.uwaterloo.ca/old/onlinetools/airprop/airprop.html)				
ρ	1023	kg/m³		
μ	8.62E-04	kg/m·s		
Cp	3420	j/kg·K		
k	0.466	W/m·K		
Pr	6.3	-		
Re	**8,000**	(assumed that flow velocity in the tubes is adjusted to achieve this Reynolds number)		
n	0.3	(exponent for cooling)		
Nu	**53**		Dittus-Boelter: $Nu_D = .023Re^{4/5}Pr^n$	
D	0.0235	m	(tube diameter)	
h	**1050**	W/m²·K	(Nu = h * D / k)	

End of presentation