93/100

Good work, very clear report

General comments:

You need to compare hand calculations to your simulations.

Time estimate is missing

Heat Exchanger Design for Oil Cooling in a Gasoline IC Engine using Ansys Fluent

B. Bunt, B. Lee, E. Taglia, P. Sklavounos

The Cooper Union for the Advancement of Science & Art

Abstract: A heat exchanger was designed to cool down hot motor oil at 350°F in a gasoline internal combustion engine under the specified maximum load conditions (25 mph moving speed through air at 108°F). The design goal was to reduce the temperature of the oil to 215°F, while maintaining realistic dimensions. This was achieved through a transient thermal analysis of the system using a combination of manual calculations, material analysis, and Computational Fluid Dynamics (CFD) software. The total weight of the duct and heat exchanger is 266.95 lb.

Design Statement: A client has commissioned our team to design a compact heat exchanger for a Gasoline IC engine to cool hot oil that will minimize costs. The final temperature of the cooled oil must at most 215°F.

The design features a high surface area-to-volume ratio, enabling effective heat transfer between the hot oil and the cooling medium, which is air. Additionally, the heat exchanger features an aluminum shell and copper tubing, both metals with high thermal conductivities.

Design Benefit: The heat exchanger system outlined in this report can cool down engine oil at 350°F to 215°F so that it may be reused to lubricate moving parts in a car's engine without overheating.

Design Constraints & Assumptions:

- Air temperature is 108°F, at a minimum of 25 mph vehicle speed
- Inlet face less than 36" in width or height, face area equal or less than 300 in^2
- Heat exchanger mounted 60" downstream of inlet of supply duct due to packaging constraints
- Length must be realistic compared to a typical street vehicle
- The inlet to the duct is offset 24" to the side of the heat exchanger
- The flow rate of oil at maximum operating load is to be between 4.5 gpm and 6 gpm
- Oil's SG = 0.86
- Oil's dynamic viscosity, μ = 34.5 cP
- Oil's thermal conductivity k = 0.15 Btu/hr.-ft- ° R
- C_p= 0.5 Btu/lbm- °R
- Forced flow conditions (pump: oil and vehicle speed: air)
- Separated flow design

I. Materials

Fluids

Air was used as the colder fluid and its properties from Ansys Fluent's material database were used (Table 1).

Table 1: Air Properties

Air Density (lbm/ft^3)	Air Viscosity (lb/ft s)	Specific Heat (Btu/lbm R)	Thermal Conductivity (Btu/h ft R)
0.076	1.20e-05	0.240	0.014

Motor oil is the hotter fluid, and its properties were provided by the design assumptions & requirements.

Oil Specific Gravity	Oil Dynamic Viscosity (cP)	Specific Heat (Btu/lbm R)	Thermal Conductivity (Btu/h ft R)
0.86	34.5	0.5	0.15

Solids

Aluminum was chosen for the shell of the heat exchanger due to its benefits as a lightweight, electrically non-conductive metal with high thermal conductivity. Copper was chosen for the oil piping material inside the heat exchanger, as it has one of the highest thermal conductivities among metals. This would maximize the heat transfer from the hotter oil to the colder air. Copper's properties were provided by Ansys Fluent's material database (Table 2). Polypropylene was chosen for the air duct material given its light density and insulative properties (Goncalves and Margarido, 2015, British Plastic Foundation). An insulative material was chosen for the duct as it is important to maintain air temperature as it travels through the duct.

Table 2: Thermal Properties of Solid Materials

Material	Density (lbm/ft^3)	Specific Heat (Btu/lbm- °R)	Thermal Conductivity (Btu/h-ft- °R)
Copper	560.5	0.091	224.0
Aluminum 6061	168.7	0.208	116.97
Polypropylene	56.997	0.459	0.116

Heat Exchanger Design

To better cool the oil, two instances of the cylindrical design pictured below were considered to decrease the mass flow rate of the oil (per individual heat exchanger) and therefore achieve prolonged heat transfer and contact between colder air and hotter oil.

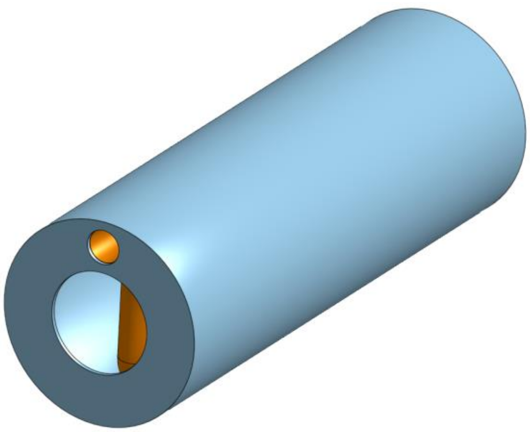


Figure 1: Solid exterior view of heat exchanger shell, with small inlet at top for oil and larger inlet at center for air (copper tubes are shown inside).

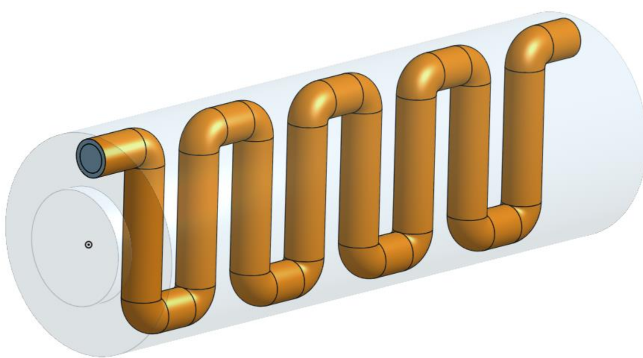


Figure 2: Transparent view of heat exchanger tubes/pipes for oil

II. Heat Exchanger Dimensions

All dimensions are in inches.

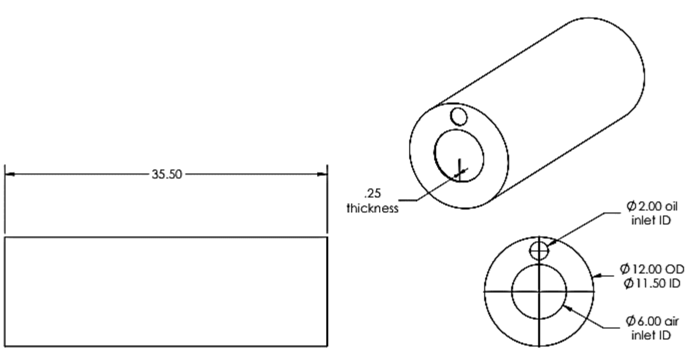


Figure 3: Dimensions for heat exchanger shell (plus 1 inch of air inlet and air outlet)

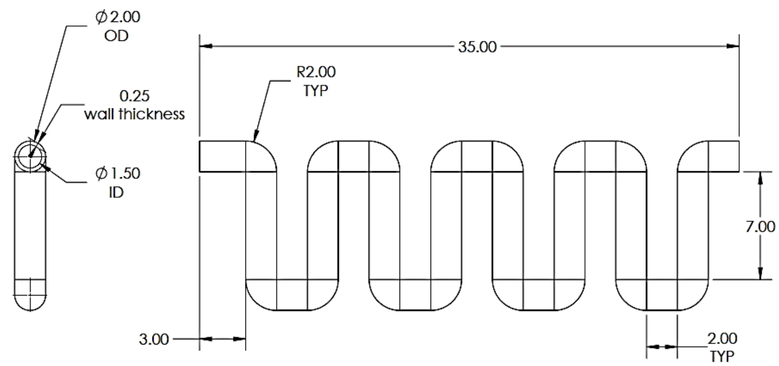


Figure 4: Dimensions for heat exchanger oil tubes

With two cylindrical heat exchanger instances, the overall width of the heat exchangers is approximately 27 inches, which meets the criteria.

III. Air Duct Design

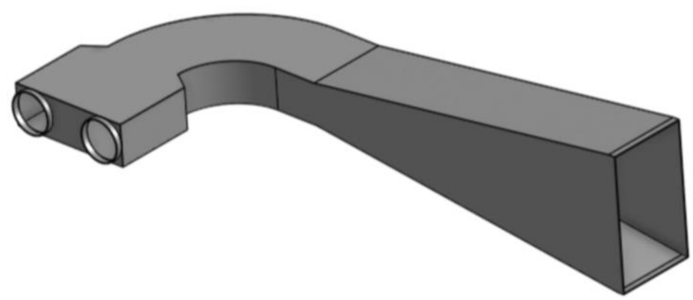
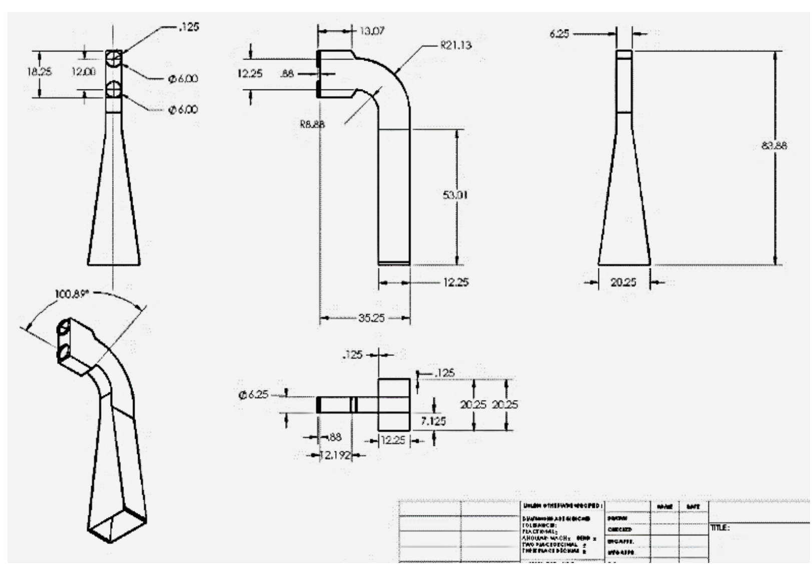


Figure 5: Solid exterior view of air duct, with intake at right and outlets (air inlets for heat exchangers) at left

IV. Air Duct Dimensions in inches



V. Hand Calculations

a. Entrance and exit temperatures of air and oil

Using the Energy Balance Equation:

$$m_{oil} \cdot c_{p_{hot}} \cdot (T_{in_{oil}} - T_{out_{oil}}) = m_{air} \cdot c_{p_{air}} \cdot (T_{out_{air}} - T_{in_{air}})$$

Rearranging, we can solve for the outlet temperature:

$$T_{out_{air}} = T_{in_{air}} + \frac{(m_{oil} \cdot c_{p_{oil}} \cdot (T_{in_{oil}} - T_{out_{oil}}))}{(m_{air} \cdot c_{p_{air}})}$$

$$T_{out_{air}} = 145 \text{ °F}$$
$$T_{in_{air}} = 70\text{°F}$$

Using the given specific heat capacity, we can calculate the heat transfer rate:

$$Q = m_{dot} \cdot c_p \cdot (T_{out} - T_{in})$$
$$Q = 0.54 \cdot 0.5 \cdot (215 - 350)$$
$$Q = -36.27 \frac{BTU}{s}$$

Using the given surface area of 300 square inches, we calculate the overall heat transfer coefficient:

$$U = 500 \cdot \left(\frac{300}{12}\right)$$
$$U = 12{,}500 \frac{BTU}{hr \cdot R}$$

Using the logarithmic mean temperature difference method, we approximate the outlet temperature of oil as:

$$LMTD = (T_1 - T_2) \ln\left(\frac{T_2}{T_1}\right)$$
$$LMTD = \frac{(350 - 215)}{\ln\left(\frac{350}{215}\right)}$$

$$T_2 = T_1 - \left(\frac{Q}{U \cdot A \cdot LMTD}\right)$$
$$T_2 = 214.9°F$$

b. <u>Pressure drop for the oil flow</u>

We use the Darcy-Weisbach equation to approximate the pressure drop of the oil flow:

$$\Delta P = f \cdot \left(\frac{L}{D}\right)\left(\rho \cdot \frac{V^2}{2}\right)$$

Assuming a fully developed flow, we can use the Colebrook equation to calculate the friction factor:

$$\frac{1}{\sqrt{f}} = -2.0 \cdot \log 1\,0\left(\left(\frac{\epsilon}{3.7 \cdot D}\right) + \frac{2.51}{Re\sqrt{f}}\right)$$

Where E is the roughness height of the pipe, and Re is the Reynolds number:

Assuming a commercial steel pipe with a diameter of 1 inch, and an oil viscosity of 34.5 cP, we calculate the Reynolds number:

$$Re = \frac{\rho V D}{\mu}$$

Re = 6055.5

Using this value, we can solve the Colebrook equation iteratively to get a friction factor of

F = 0.0217

Thus, using the Darcy-Weisbach equation, the approximate pressure drop of the oil flow is 1.26 psi.

c. <u>Efficiency of the heat exchanger</u>

The $\epsilon - NTU$ method will be used for reasons outlined in Discussion.

The efficiency of a heat exchanger is given by a ratio of the volume of heat transferred between the two fluid streams in the exchanger to the maximum possible heat transfer that could occur under ideal conditions:

$$\varepsilon = \left(\frac{Q_{actual}}{Q_{max}}\right)$$

Where,

$$Q_{actual} = \dot{m}_{hot} \cdot c_{p_{hot}} \cdot (T_{hot_{in}} - T_{hot_{out}})$$
$$Q_{actual} = 2.76 \cdot 0.24 \cdot (350 - 108)$$
$$Q_{actual} = 26.71 \frac{ft^3}{s}$$

and

$$Q_{max} = \dot{m}_{min} \cdot c_{p_{hot}} \cdot (T_{hot_{in}} - T_{cold_{out}})$$
$$Q_{max} = 2.76 \cdot 0.24 \cdot (350 - 108)$$
$$Q_{max} = 2.76 \cdot 0.24 \cdot (350 - 80)$$
$$Q_{max} = 29.808$$
$$\eta = \left(\frac{Q_{actual}}{Q_{max}}\right)$$
$$\eta = \left(\frac{26.71}{29.808}\right) = 89.1\% \; efficient$$

Further hand calculations are found in Appendix IX

VI. Entire Design Assembly

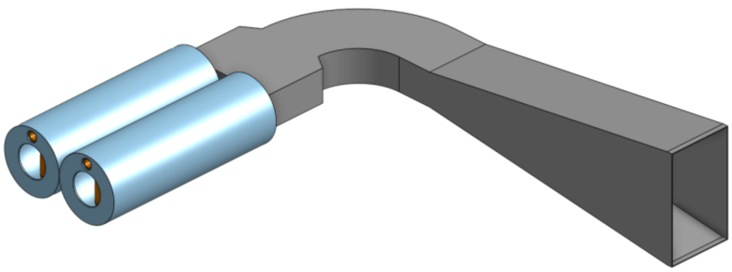


Figure 6: Isometric View of the Assembly (air duct and heat exchanger)

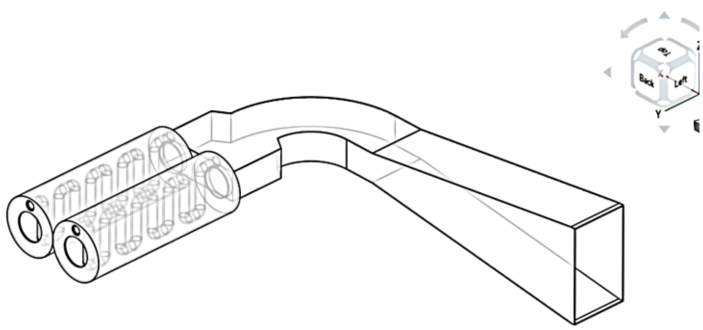


Figure 7: Isometric, Transparent View of the Assembly (air duct and heat exchanger)

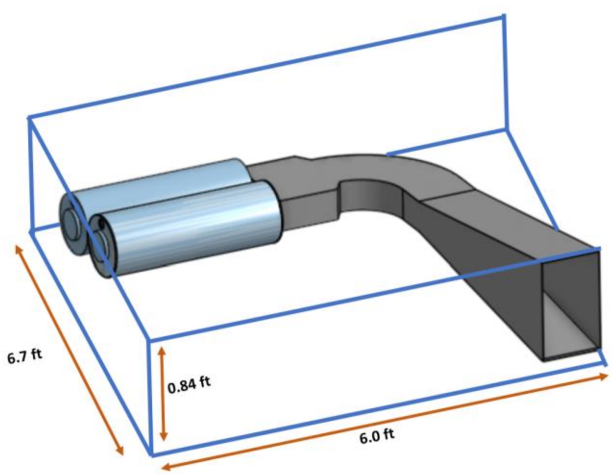


Figure 8: Black box dimensions of duct and heat exchanger

Table 3: Dry weight of heat exchanger parts

Part	Dry Weight (lb)
Copper Tube X2	89.0
Polypropylene air duct	18.9
Shell X2	70.0
Total	266.05

VII. Meshing

ANSYS Meshing was used for meshing the duct model. An average element size of 0.787 inches was used for the duct, with inflation and layers used around areas of interest, such as bends and openings (inlet and outlet).

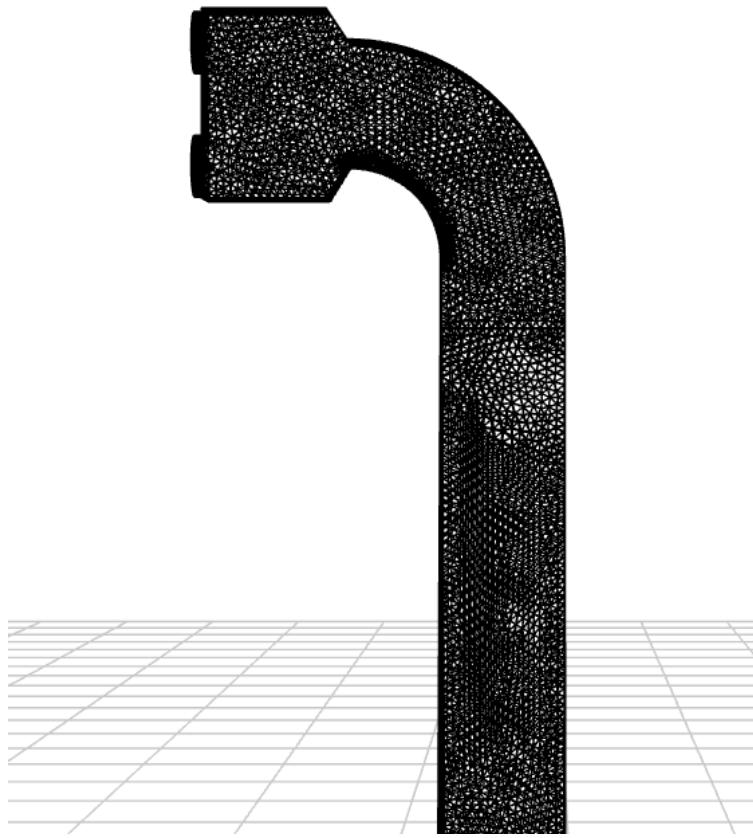


Figure 9: Overall Mesh of Duct

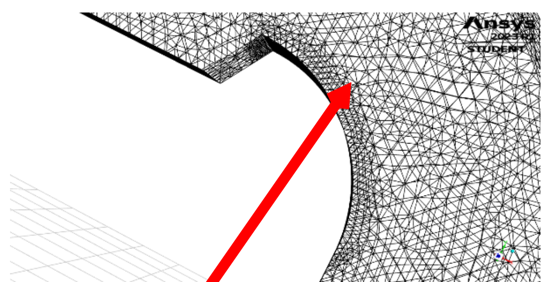


Figure 10: Neck of Duct Mesh – Inflation Detail

Filled images are easier to read

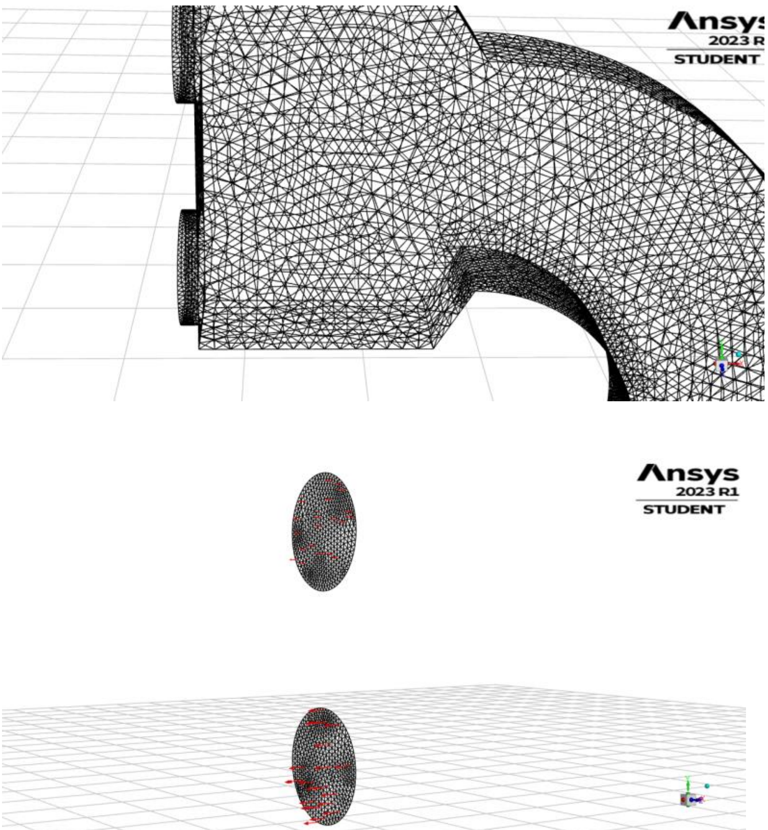


Figure 11: Duct Outlet Mesh Details

ANSYS Meshing was also used for meshing the heat exchanger model. To simplify meshing and the CFD analysis of the model, the model was split into half. A tetrahedral mesh was generally applied throughout the zone of colder air, while a hexahedral mesh was generally applied throughout the zone of hotter oil. Body sizing was used for the oil and the air, with an element size of 0.1875in and 0.375in, respectively. For the bends, rather than using an element size, the number of divisions for the outer bend were defined at 15 divisions.

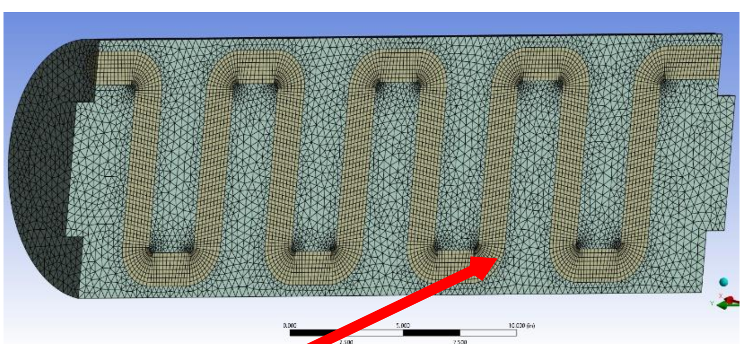


Figure 12: Mesh used for heat exchanger

Swpt meshes may not be ideal for tight turns as the aspect ratio of the elements may become skewed

Resolution is a bit coarse here. Minimum 6 cells in gaps is required.

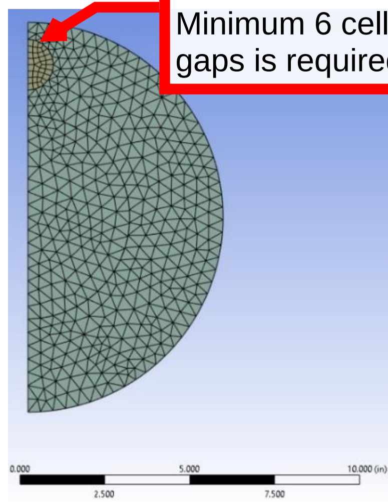


Figure 13: Mesh up close

The following mesh statistics were obtained.

Quality Criterion	Warning (Target Limit)	Error (Failure Limit)	Worst Value Noted
Minimum Element Quality	0.05	0.0005	0.211
Maximum Aspect Ratio	5	1000	10.198
Minimum Orthogonal Quality	0.05	0.005	0.201
Maximum Skewness	0.9	0.999	0.799
Minimum Tet Collapse	0.1	0.001	0.199

The minimum element quality measures the shape of each element, and a low value indicates that the elements are deformed. The maximum aspect ratio measures the elongation of each element, and a high value indicates that the elements are stretched. The minimum orthogonal quality measures how orthogonal the faces of each element are, and a low value indicates that the faces are not perpendicular. The maximum skewness measures how much the elements deviate from being perfectly symmetrical, and a high value indicates that the elements are distorted. The minimum tetrahedral collapse measures the collapse of tetrahedral elements in the mesh, and a low value indicates that the tetrahedra are collapsing.

To reduce the likelihood of a severe tetra-collapse (in which the tetra collapse value of one or more elements is below 0.2), especially for thin geometries such as the shell of the heat exchanger and the 1/8-inch-thick copper piping in the heat exchanger, the oil piping was excluded from meshing. A thickness of 0.125 inch (around 0.0104 foot) was defined in Ansys Fluent for a contact region subsequently created between the hotter oil and the cooler air (Figure 14).

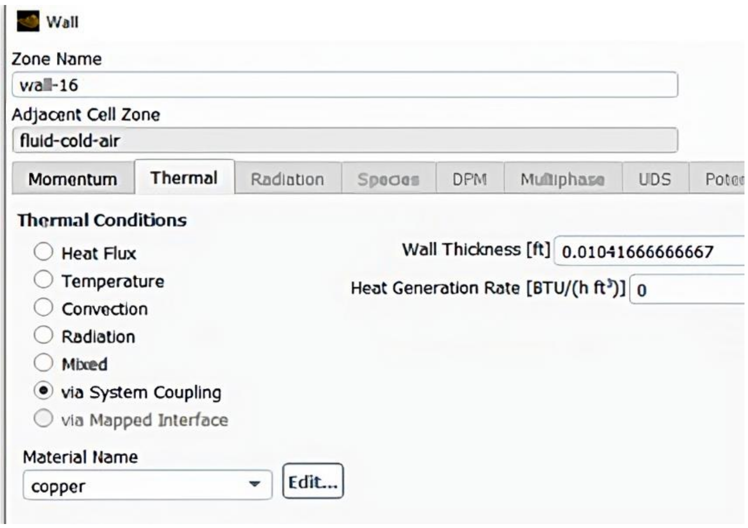


Figure 14: Wall Thermal Conditions for Shell in Ansys Fluent submenu

VIII. Simulation Results

Numerous simulations were conducted with one of the heat exchangers, as well as the air duct leading to the heat exchangers.

All quantities for oil in the heat exchanger were known. However, the temperature and speed of the air after it has traveled inside the duct (from the intake at the front of the car) needed to be determined. This was accomplished by running simulations of the air duct using an initial speed of 25 mph and an initial temperature of 108 ˚F.

The realizable k-epsilon turbulence model was used with appropriate boundary conditions. More information on the simulation setup is provided in Appendix c.

The following area-weighted velocity averages were obtained.

Area-Weighted Average Velocity Magnitude	[mph]
Closer Outlet	96.604
Further Outlet	136.200
Average	116.402

The duct outlet speeds calculated above were used as the air inlet into the heat exchanger. The realizable k-epsilon turbulence model was used with appropriate boundary conditions. 900 iterations were requested.

a. Duct Simulations (25 mph Inlet)

Figure 15 shows a temperature contour plot across the midplane. It was determined that the temperature stays constant throughout the duct given the small variations in temperature that are negligible.

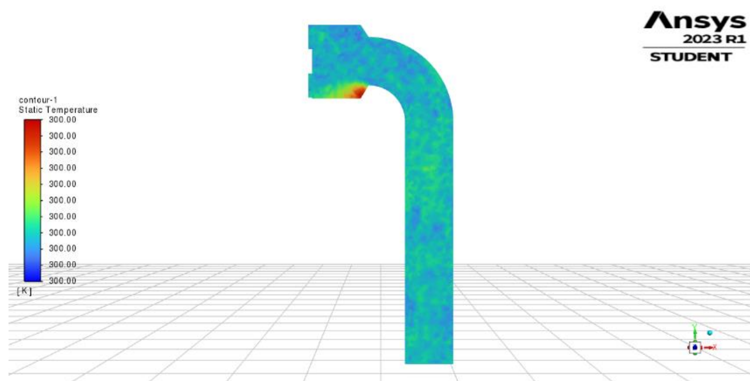


Figure 15: Temperature contour plot of almost constant temperature

Figure 16 shows the pressure contour of the duct. The red portions are where the pressure is around 0.6 psi, and the blue portions are around –0.02 psi.

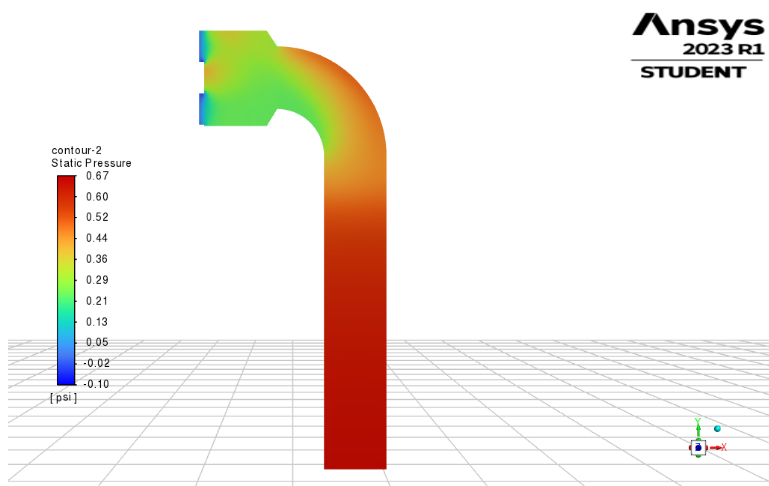
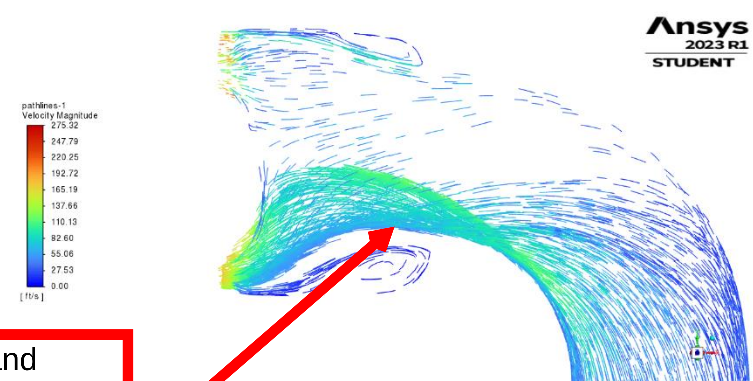


Figure 16: Pressure Contour throughout Duct

Figures 17 & 18 illustrate the velocity streamlines and velocity vectors, respectively. They visualize where the velocity increased throughout the duct. The values from these simulations were consequently used as the inlet velocity of air for the heat exchanger.



Use arrows and fewer streamlines to make this clearer

Figure 17: Duct Velocity Magnitude Streamline Plot

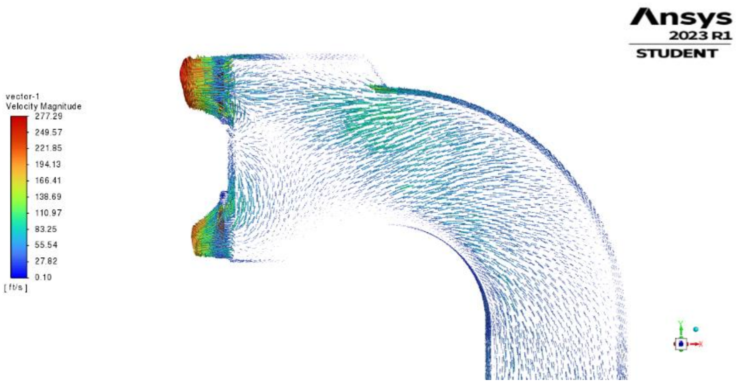


Figure 18: Duct Velocity Magnitude Vector Plot

b. <u>Heat Exchanger 1 (96.6 mph Inlet)</u>

Figure 19 shows residuals as plotted by Ansys Fluent. They are noted to be converging. Figure 20 verifies that a steady state is indeed reached.

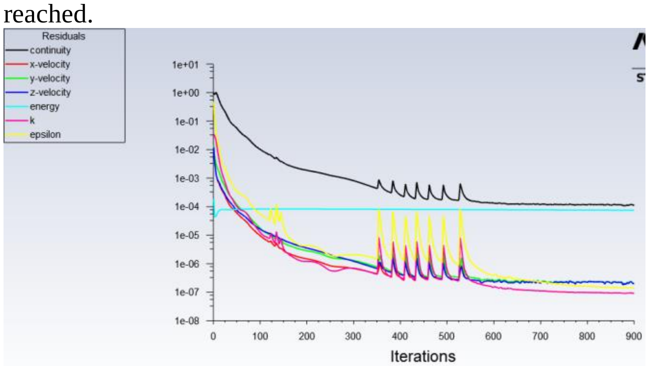


Figure 19: Residual Convergences for Simulation of Heat Exchanger

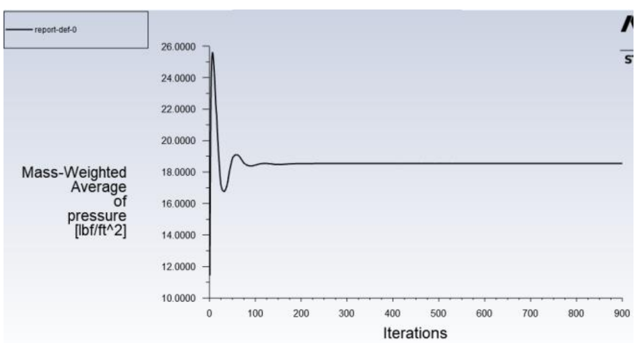
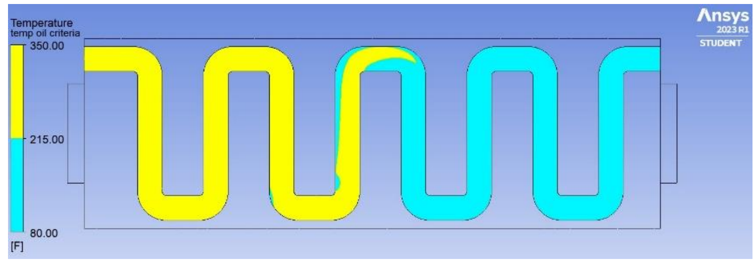
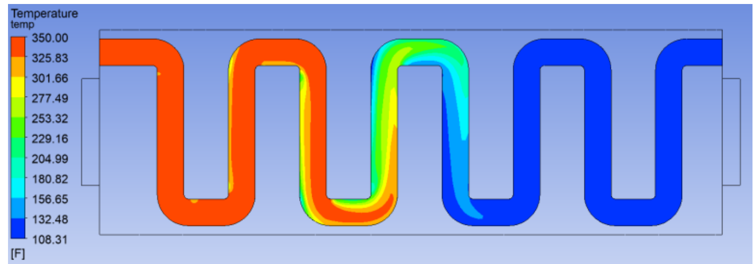


Figure 20: Sample Steady-State Plot of Oil Temperature

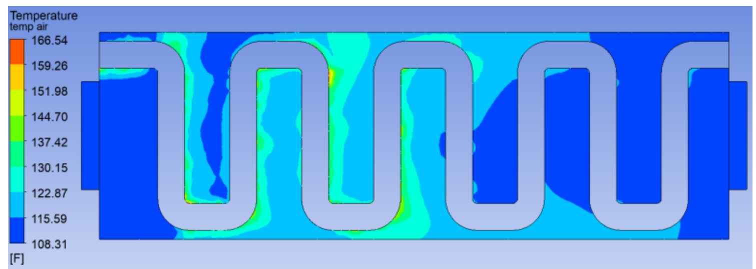
Areas where the oil cools to 215 ˚F or lower (blue portion) are shown below, with the left side being the inlet of oil and the right side being the outlet of oil. Yellow portions are where the oil is hotter than 215 ˚F.

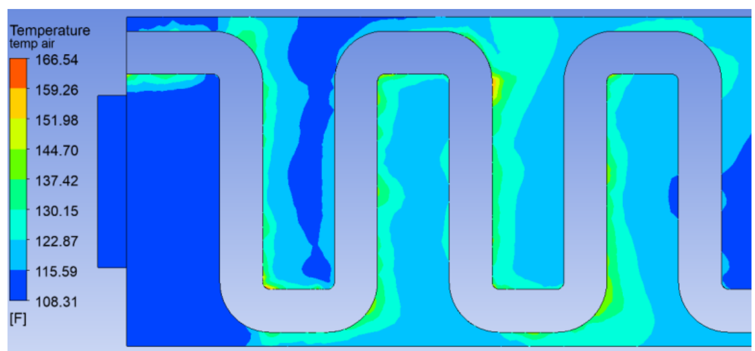


More detailed temperature distribution results of the oil are displayed below.



The temperature distribution of the air throughout the heat exchanger, and a closer image of hottest air temperatures are displayed below.



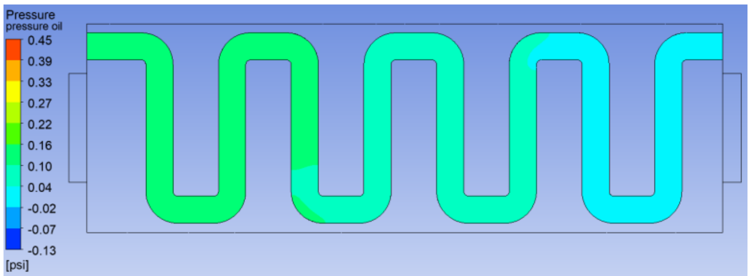


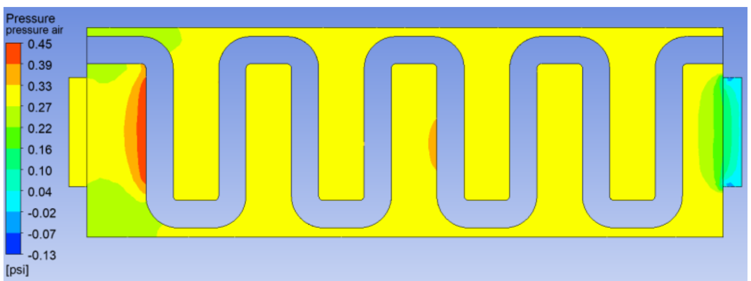
The following area-weighted temperature averages (at inlets and outlets) were obtained after the 900 iterations.

Area-Weighted Average Static Temperature	Temp [°F]
Inlet Air	108.330
Outlet Air	111.027
Inlet Oil	350.0
Outlet Oil	122.269

Watch significant figures

Pressure contours for the oil and air (respectively) are shown below.

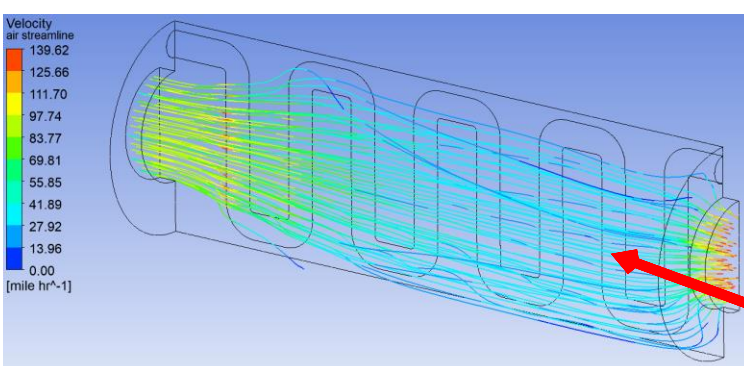




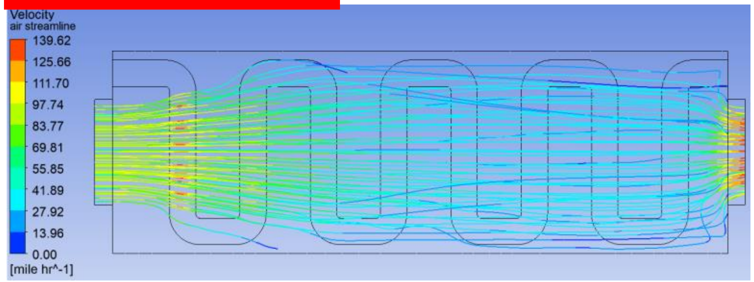
The following area-weighted gage pressure averages (at inlets and outlets) were obtained after the 900 iterations.

Area-Weighted Average Static Pressure	[psi]
Air Inlet	0.565
Air Outlet	-0.00274
Oil Inlet	0.152
Oil Outlet	0

Velocity streamline plots for the air are shown below.



Use thicker and fewer streamlines



c. <u>Heat Exchanger Case: 2 (136.2 mph Inlet)</u>

Figure 21 shows residuals as plotted by Ansys Fluent. They are noted to be converging. Figure 22 verifies that a steady state is reached.

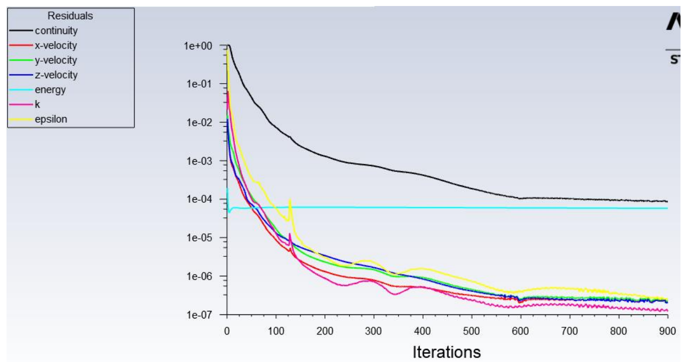


Figure 21- Residual Convergences for Simulation of Heat Exchanger

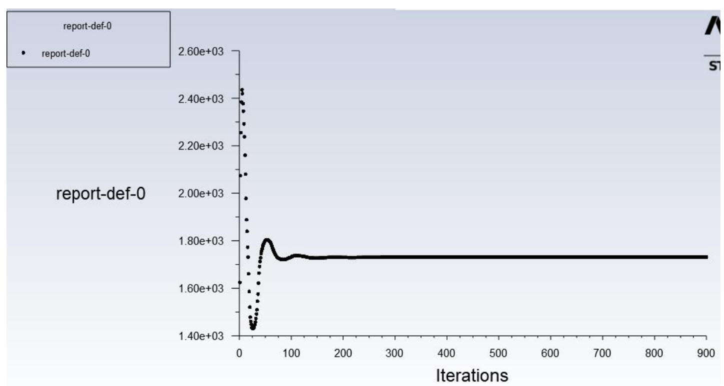
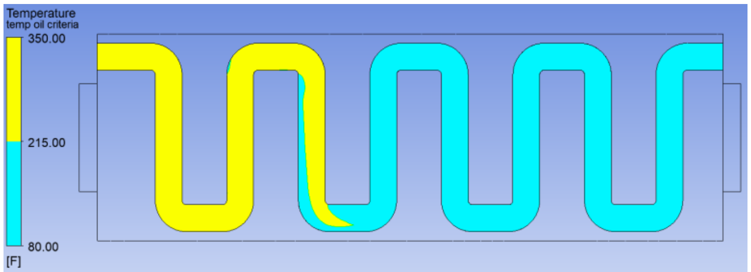
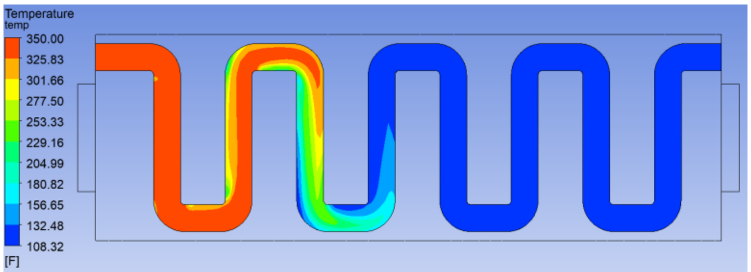


Figure 22: Sample Steady-State Plot of Variable

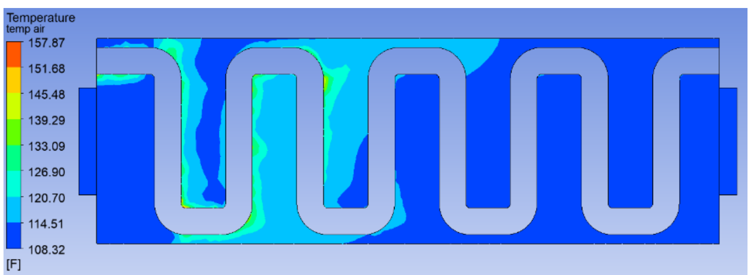
Areas where the oil cools to 215 °F or lower (blue portion) are shown below, with the left side being the inlet of oil and the right side being the outlet of oil. Yellow portions are where the oil is hotter than 215 °F.



More specific temperature distribution results of the oil are displayed below.



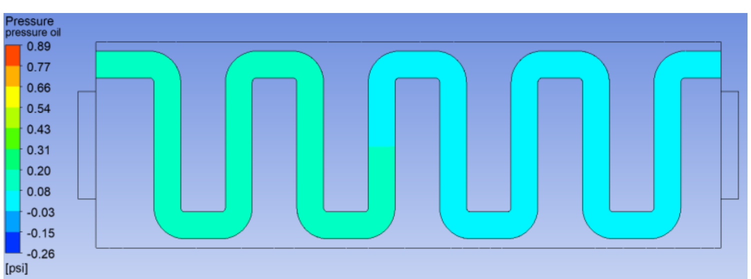
Temperature distribution results of the air are displayed below.

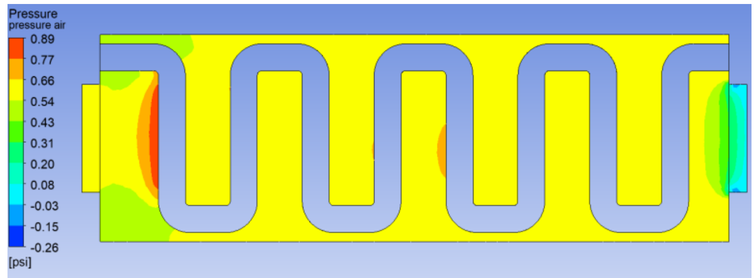


The following area-weighted temperature averages (at inlets and outlets) were obtained after the 900 iterations.

Area-Weighted Average Static Temperature	[˚F]
Air Inlet	108.330
Air Outlet	110.032
Oil Inlet	350.00
Oil Outlet	122.288

Pressure contours for the oil and air (respectively) are shown below.

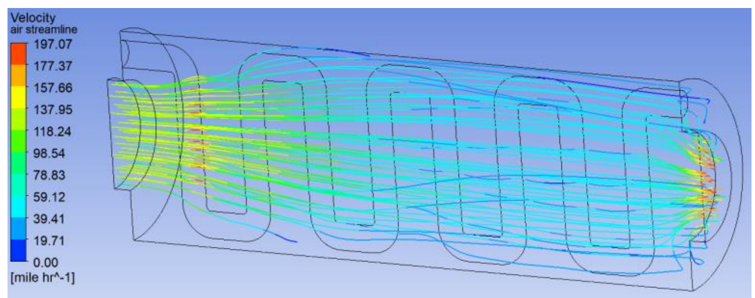


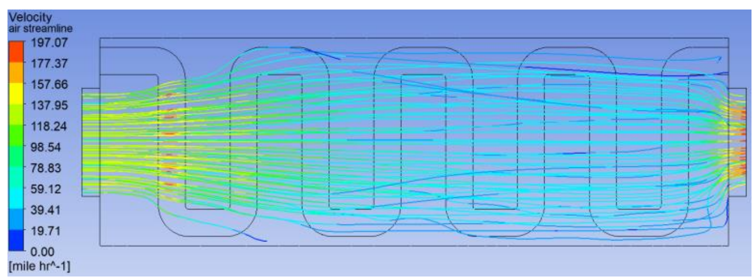


The following area-weighted gage pressure averages (at inlets and outlets) were obtained after the 900 iterations.

Area-Weighted Average Static Pressure	[psi]
Air Inlet	0.565
Air Outlet	-0.00274
Oil Inlet	0.152
Oil Outlet	0

Velocity streamline plots for the air are shown below.





IX. Discussion

Based on the temperature contour of the oil in Section VIII and the oil outlet temperature noted above, the heat exchanger design can meet the criteria of cooling engine oil from a temperature of 350 ˚F to maximum 215 ˚F.

To calculate the efficiency of the heat exchanger, the $\epsilon - NTU$ method is used, where the volume of heat transferred between the two fluid streams in the exchanger is compared to the maximum possible heat transfer that could occur under ideal conditions. This is known as the efficiency of the heat exchanger which is given by a ratio of these heat transfer amounts:

$$\eta = (\frac{Q_{actual}}{Q_{max}})$$

Variable notation	Variable meaning
Q_{actual}	Actual heat transfer that occurs in the heat exchanger
Q_{max}	The maximum possible heat transfer that could occur if the temperature difference between the two fluid streams was equal to the temperature difference between the hot and cold fluid streams.

To calculate Q_{actual}, the inlet and outlet temperatures of the hot and cold fluid streams are used to calculate the heat transfer rate using the specific heat capacity equation:

$$Q_{actual} = \dot{m}_{hot} \cdot c_{p_{hot}} \cdot (T_{hot_{in}} - T_{hot_{out}})$$

Variable notation	Variable meaning
$\dot{m}_{hot}$	Mass flow rate of the hot fluid.
$c_{p_{hot}}$	Specific heat capacity of the hot fluid.
$T_{hot_{in}}$	The inlet temperature of the hot fluid.
$T_{hot_{out}}$	The outlet temperature of the hot fluid.

Similarly, the heat transfer for the cold fluid is calculated. The smallest heat transfer rate is chosen and compared to Q_{max}, which is the maximum possible heat transfer rate being the rate that would occur if the two fluids were completely mixed:

$$Q_{max} = \dot{m}_{min} \cdot c_{p_{min}} \cdot (T_{hot_{in}} - T_{cold_{out}})$$

For our given context, the NTU method is relevant as we can quickly find the efficiency using only inlet and outlet temperatures without using iterative numerical methods with nonlinear equations, and the fluids used are single phased [2][3].

Advantages	Disadvantages
Simple and easy to use for the design and analysis of heat exchangers.	Assumes a constant heat transfer coefficient and fluid properties, which may not be accurate in some cases.
Can be applied to a wide range of heat exchanger types, including both parallel and counter-flow configurations.	Limited to single-phase fluids and cannot be used for condensing or boiling heat transfer.
Allows for quick estimation of the heat exchanger effectiveness and performance.	Does not provide detailed information about the temperature and flow distribution within the heat exchanger.
Can be used to optimize the heat exchanger design for different operating conditions.	May not be accurate for heat exchangers with very low or very high effectiveness.
Can be easily incorporated into spreadsheet programs for rapid analysis of heat exchanger designs.	

X. Conclusion

The design of the heat exchanger and duct proved to lower the temperature of the oil, using incoming air, to the required temperature while remaining compact and realistic. The use of copper tubes with a surface area to volume ratio of 2.86, an aluminum shell heat exchanger, and a polypropylene duct were used given their several advantages. Copper has excellent thermal conductivity, making it a suitable material for heat exchanger tubes, while aluminum is lightweight, making it ideal for the shell of the heat exchanger. Polypropylene is a durable and cost-effective material that can withstand high temperatures and corrosive substances, making it a good choice for the air duct. Overall, this design offers a practical and efficient solution for cooling hot motor oil, which can lead to better performance and longer lifespan of the motor.

XI. Appendix

a. Hydraulic Diameter [4]

$$D_h = 4 \cdot (\frac{A}{P}) \text{ where A}$$

Variable notation	Variable meaning
D_h	Hydraulic diameter
A	The duct or pipe area
P	Perimeter of the duct or pipe cross-section.

Did you account for radiation heat transfer

b. Boundary Conditions of Heat Exchanger

Boundary Conditions

− Inlet	
− inlet-oil	
Reference Frame	Absolute
Mass Flow Specification Method	Mass Flow Rate
Mass Flow Rate [lbm/s]	0.2690187
Total Temperature [R]	809.67
Supersonic/Initial Gauge Pressure [lbf/ft^2]	0
Direction Specification Method	Normal to Boundary
Turbulent Specification Method	Intensity and Hydraulic Diameter
Turbulent Intensity [%]	5
Hydraulic Diameter [ft]	0.01041667
Note: Reinjected particles do not change their injection association	-2112242544
− inlet-air	
Velocity Specification Method	Magnitude, Normal to Boundary
Reference Frame	Absolute
Velocity Magnitude [mph]	96.60381
Supersonic/Initial Gauge Pressure [lbf/ft^2]	0
Temperature [R]	568
Turbulent Specification Method	Intensity and Hydraulic Diameter
Turbulent Intensity [%]	5
Hydraulic Diameter [ft]	0.5
Note: Reinjected particles do not change their injection association	1954525584
− Outlet	
− outlet-air	
Gauge Pressure [lbf/ft^2]	0
Pressure Profile Multiplier	1
Note: Reinjected particles do not change their injection association	455645760
Build artificial walls to prevent reverse flow?	yes
Radial Equilibrium Pressure Distribution	no
Average Pressure Specification?	no
Specify targeted mass flow rate	no
− outlet-oil	
Gauge Pressure [lbf/ft^2]	0
Pressure Profile Multiplier	1
Note: Reinjected particles do not change their injection association	461476080
Build artificial walls to prevent reverse flow?	yes
Radial Equilibrium Pressure Distribution	no
Average Pressure Specification?	no
Specify targeted mass flow rate	no
− Symmetry	
symmetry-fluid-hot-oil	symmetry
symmetry-fluid-cold-air	symmetry
− Wall	
− wall-6-shadow	
Wall Thickness [ft]	0
Heat Generation Rate [BTU/(h ft^3)]	0
Material Name	copper
Thermal BC Type	Coupled
Enable shell conduction?	no
Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip
Convective Augmentation Factor	1
− wall-6	
Wall Thickness [ft]	0
Heat Generation Rate [BTU/(h ft^3)]	0
Material Name	copper
Thermal BC Type	Coupled
Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip
Convective Augmentation Factor	1
− wall-17	
Wall Thickness [ft]	0.01041667
Heat Generation Rate [BTU/(h ft^3)]	0
Material Name	copper
Thermal BC Type	via System Coupling
Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip
Convective Augmentation Factor	1
− wall-shell	
Wall Thickness [ft]	0
Heat Generation Rate [BTU/(h ft^3)]	0
Material Name	aluminum
Thermal BC Type	Heat Flux
Heat Flux [BTU/(h ft^2)]	0
Enable shell conduction?	no
Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip

Use graphics with this to show what each boundary is

c. Model of Heat Exchanger Simulation

Models

Model	Settings
Space	3D

Model	Settings
Time	Steady
Viscous	Realizable k-epsilon turbulence model
Wall Treatment	Enhanced Wall Treatment
Heat Transfer	Enabled

d. Model and Boundary Conditions of Duct

Models

Model	Settings
Space	3D

Model	Settings
Time	Steady
Viscous	SST k-omega turbulence mode
Heat Transfer	Enabled

<u>Boundary conditions</u>

− Inlet	
− Inlet	
Velocity Specification Method	Magnitude, Normal to Bounda
Reference Frame	Absolute
Velocity Magnitude [ft/s]	37.72956
Supersonic/Initial Gauge Pressure [lbf/ft^2]	0
Temperature [R]	540
Turbulent Specification Method	Intensity and Hydraulic Diame
Turbulent Intensity [%]	5
Hydraulic Diameter [ft]	0.1245719
Note: Reinjected particles do not change their injection association	no
− Outlet	
− close_outlet	
Backflow Reference Frame	Absolute
Gauge Pressure [lbf/ft^2]	0
Pressure Profile Multiplier	1
Backflow Total Temperature [R]	540
Backflow Direction Specification Method	Normal to Boundary
Turbulent Specification Method	Intensity and Hydraulic Diameter
Backflow Turbulent Intensity [%]	5
Backflow Hydraulic Diameter [ft]	0.5
Note: Reinjected particles do not change their injection association	no
Backflow Pressure Specification	Total Pressure
Build artificial walls to prevent reverse flow?	no
Radial Equilibrium Pressure Distribution	no
Average Pressure Specification?	no
Specify targeted mass flow rate	no
− far_outlet	
Backflow Reference Frame	Absolute
Gauge Pressure [lbf/ft^2]	0
Pressure Profile Multiplier	1
Backflow Total Temperature [R]	540
Backflow Direction Specification Method	Normal to Boundary
Turbulent Specification Method	Intensity and Hydraulic Diameter
Backflow Turbulent Intensity [%]	5
Backflow Hydraulic Diameter [ft]	0.5
Note: Reinjected particles do not change their injection association	no
Backflow Pressure Specification	Total Pressure
Build artificial walls to prevent reverse flow?	no
Radial Equilibrium Pressure Distribution	no
Average Pressure Specification?	no
Specify targeted mass flow rate	no
− Wall	
− wall-part_1	
Wall Thickness [ft]	0
Heat Generation Rate [BTU/(h ft^3)]	0
Material Name	aluminum
Thermal BC Type	Heat Flux
Heat Flux [BTU/(h ft^2)]	0
Enable shell conduction?	no
Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip
Wall Surface Roughness	0
Wall Roughness Height [ft]	0
Wall Roughness Constant	0.5
Convective Augmentation Factor	1

References

[1] E.A. Avallone, T. Baumeister III, and A. Sadegh. "Heat Transfer." Marks' Standard Handbook for Mechanical Engineers, 12th Edition. McGraw-Hill Education, 2017, pp. 12-1–12-100.

[2] Heat exchanger - Department of Chemical Engineering Available at: http://ww2.che.ufl.edu/ren/unit-ops-lab/ech4224L/HE/HE-theory.pdf (Accessed: April 1, 2023).

[3] F. P. Incropera, D. P. DeWitt, T. L. Bergman, and A. S. Lavine, "Fundamentals of Heat and Mass Transfer", 6th ed. (Wiley, 2007).

[4] ASHRAE Handbook of Fundamentals Chapter 21

Midterm Design Project

Report and Presentation Due: Monday, April 3rd 6:00 pm
Progress Presentation: Monday, March 20th 6:00 pm

1 Overview

You have been asked to design a heat exchanger which will be used as an oil cooler in a gasoline IC engine. This heat exchanger you will be designing will be used to ensure that the lubricating oil does not thermally breakdown. The heat exchanger will be an oil cooler with the rejecting medium being air that will be delivered via a duct. The duct and the heat exchanger comprise your device.

Your goal is to optimize the efficiency of the heat exchanger using CFD as a tool to find the optimal geometry. The primary drivers of your design will be to reduce the size and weight of the heat exchanger to keep costs reasonable.

2 Performance Requirements

Your heat exchanger design will utilize a separated flow design, whereby your hot fluid is light oil (motor oil), and your working cold fluid is air. Both fluids will be under forced flow conditions, the oil via a pump, and the air driven by the vehicle speed. The specific requirements of the project are as follows:

- The heat exchanger shall be capable of rejecting the required heat when the air temperature is 108°F and the vehicle is traveling at a minimum of 25 mph (it does not have to work below this speed).
- The heat exchanger inlet face must be less than 36" in width or height. The inlet size (face area) to the heat exchanger must be 300 in^2 or less. The device may be as deep as required.
- Due to packaging constraints, the heat exchanger is mounted 60" downstream of the inlet of a supply duct.
    - The air inlet side of the duct on the front of the vehicle has a section of 20" x 12" (height x width)
    - The duct may transition in cross section over its length, or at the outlet, but must exactly fit the inlet of the heat exchanger at the outlet location of the duct.
    - The inlet to the duct is offset 24" to the side of the heat exchanger
- The specification of oil temperature at maximum load is:
    - Entering: 350°F
    - Exiting: 215°F maximum
- The flow rate of oil at maximum operating load is to be between 4.5 gpm and 6 gpm.

2.1 Assumptions

- You may assume the oil has the following properties:
    - SG = 0.86
    - μ = 34.5 cP

- $k = 0.15$ Btu/hr-ft-°R
- $c_p = 0.5$ Btu/lbm-°R

3 Project Requirements

The project's design criteria are multifaceted. They include the following:

- Meeting the design performance requirements and limit specifications.
- Be of compact design.
- All work should be reported in I-P units.

3.1 Items of non-responsibility

You will not be responsible for engineering or reporting the following beyond common sense assumptions:

- Material compatibility / costs
- Manufacturing methods / costs

4 Deliverables

Your design needs to be presented in several ways and needs to fully convey the manner in which it works. These items should include the following:

4.1 Progress Presentation (5 minutes)

Halfway through the design period you will be required to give a brief presentation () to show progress on your project. You should present at a minimum:

- Design concept
- Initial hand calculations
- Any preliminary simulations or models

4.2 Final Report

The report should be typed and submitted on time by the final delivery date and time. Your report should be of a professional level meeting the standards discussed in class and shall be submitted in electronic form (PDF). In general color or black and white images are acceptable as long as they are legible and clearly explain the results.

Your final report should include:

- Design assumptions.
- Statement of design concept.
- Scaled sketches illustrating the conceptual design.
  - The basic design of each of your device should be illustrated in clear scaled sketches. All components used should be sized and called out in your drawings. Do the anticipated geometric complexity of your components the use of 3D CAD is strongly recommended.
  - It is not necessary to draw every view of every component that is to be designed. Drawings should only be included if they show meaningful information that cannot be illustrated on another drawing.
- CFD simulations that demonstrate the following:

- The simulated entrance and exit temperatures for both working fluids
  - Air entering the duct through exiting the heat exchanger
  - Oil entering and exiting the heat exchanger
- The simulate pressure drop for both fluids across the overall device
  - Air pressure drop across the duct and heat exchanger
  - Oil pressure drop across the heat exchanger
- Temperature contour plots showing fluids under steady state conditions across the device
- Pressure contours showing the distribution of both working fluids under steady state conditions
- Velocity / streamline plots that demonstrate the flow patterns through the heat exchanger

- Hand verification calculations for the major elements of your design, including:
  - Approximations for entrance and exit temperatures for the oil and the air based on fluid, heat transfer and thermodynamic theory
  - Approximations for pressure drop for the oil flow
- The operating conditions of your design at the specified conditions, including:
  - Oil flow rate
  - Summarized overall temperature difference for both working fluids across the entire device
  - Summarized overall pressure drop for air and oil sides
- The efficiency of the heat exchanger based on simulation results
- Overall size of the unit, length, width and depth
- A dry weight estimate
- Design time estimate, cumulative number of hours spend developing your design
- Statement of design benefits.

4.3 Final Presentation (8 minutes)

Your final presentation should provide the audience with an overall summary of your design and the key analyses that were used to arrive at it. This should include:

- The design concept
- Your heat exchanger performance, i.e. temperature differentials, pressure drops, efficiency, size weight, etc.
- How CFD simulations helped influence your design
- Design time estimate
- Design benefits.

I don't see it

5 Suggestions

The following is a list of suggestions that may be helpful in getting a jump start on your design submission.

5.1 Basic design

Start with the big picture and work your way down to the details. Once you have a basic design concept the details will be much easier to formulate. CFD models are very good at not only determining detailed flow parameters for a final design but also at providing general system behavior.

5.1.1 Material properties

You may use material properties from a text book, Mark's Handbook or Fluent's material library. Check to see that the figures you have make sense.

5.1.2 Starting points

There are many resources out there for how heat exchangers work. Use them to get a basic idea as to who you want to design your system. Make sure the design is realistic; i.e. material properties used are reasonable and within normal limits. The following are recommended readings to get you started in your work:

- http://en.wikipedia.org/wiki/Heat_exchanger

- Mark's Standard Handbook for Mechanical Engineers – Section 4.4 heat transfer via conduction and convection (10th Ed.)

5.2 Modeling in general

Use the modeling tools at your disposal to your advantage. Use symmetry, sub-modeling and other modeling techniques to reduce your efforts and make a simplified model. If you can model something simply and prove your concept works that will be sufficient assuming you can back it up. Large very complicated models used upfront are often a very inefficient way of getting to a good solution.

Be aware of the limitations we have discovered in class and through assignments. Be critical of the results you generate. Check all simulations with a basic hand calculation.

5.3 Presentation of results

As in the rest of your careers you will find the best design will never be chosen if the person who is buying it does not understand or trust it. Try and make all the hard work you put into you thoughts about how to solve this problem as clear as possible. Use images and schematics to illustrate things and put time into making graphics clear and presentable.

5.4 Time

It is strongly suggested that you begin work on your design early. Design work coupled with CFD analysis can be time intensive. Ensure that you will have sufficient time to properly prepare your presentation and report.