IC Engine Lubricating Oil Heat Exchanger Design

Gary Go
Andrew Kim
Haoran (Harry) Wei

ME 407: Computational Fluid Dynamics
Professor Scott Bondi

The Cooper Union for the Advancement of Science and Art

06 April 2020

Table of Contents

Introduction

Overview

Heat exchangers are vital parts of an internal combustion (IC) engine's cooling system. It is vital for excess heat to be removed from the IC engine to prevent components from melting, pistons from seizing, and lubricating oil to thermally decompose. The objective of this IC engine heat exchanger is to protect the lubricating oil from thermal decomposition by removing enough heat from the oil via forced air cooling. The system under inspection consists of a duct to transport cooler air across a finned tube radiator that carries the hot oil through multiple passes.

IC Heat Exchanger Design and Modeling Plan

Finned tube heat exchangers consist of tubes winding through sheets of thin fins. Hot oil is pushed through the tubes, heating both the tubes and the fins. Cool air is forced through the fins and tube. The fins increase the surface area to improve heat transfer between the cool air and hot oil. A typical finned tube heat exchanger is shown below.

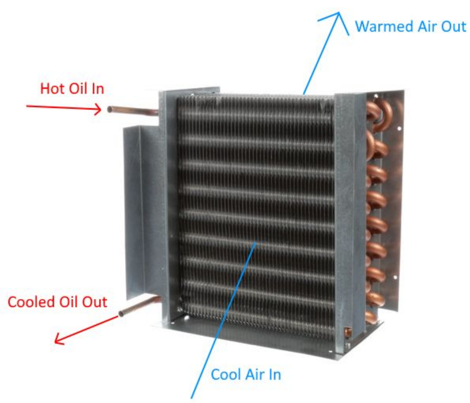


Figure 1. Finned tube heat exchanger example courtesy of webstaurantstore.com

The design of the heat exchanger system is broken down into three main design considerations: air duct shape, radiator fin parameters, and tube manifold design. The main goal of the air duct is to bring in cool air at a high speed to improve heat transfer from the oil to air. The goal of the fin parameters, which include fin thickness and spacing, is to maximize surface area for heat transfer to occur. The goal of the manifold design is to split the oil flow into multiple channels to decrease the speed of the oil, maximizing heat transfer. Each of these design aspects were simulated separately because each part should not affect the performance of the other.
Table 1 shows an outline of each model and the goal of the model.

Model Description	Model Objective
3 Air Inlet Duct Geometries	Minimize air velocity loss and pressure loss
Fin Thickness and Spacing Parametric Models	Decrease thickness and spacing without losing heat exchange performance
Tube Manifold Geometries	Decrease oil velocity via multiple channels

Table 1. The various models that simulated the three main design considerations.

Design Requirements and Assumptions

Design Constraints

Table 2, below, shows the extreme conditions and performance requirements of the IC engine heat exchanger.

Description	Requirement
Maximum Air Temperature	108°F
Maximum Oil Temperature Entering	350°F
Maximum Oil Temperature Exiting	195°F
Minimum Air Velocity	25 mph
Oil Flow Rate Range	4.0 - 5.5 gpm
Maximum Inlet Face Length of Heat Exchanger	36 in
Maximum Inlet Face Area of Heat Exchanger	300 in^2
Duct Air Inlet Face Area	20 in x 12 in (height x width)
Duct Outlet Face	Matches Heat Exchanger Inlet Face
Duct Inlet Offset from Heat Exchanger	18 in
Duct Inlet Distance to Heat Exchanger	60 in

Table 2. Design Requirements for the IC engine heat exchanger

Design Assumptions

Oil Properties

The properties of the lubricating oil is as follows:

- Specific gravity: 0.86
- Viscosity: 34.5 cP
- Thermal Conductivity: 0.15 btu/h-ft-°R
- Specific Heat: 0.5 Btu/lbm-°R

Air Properties

The properties of air is as follows:

- Density: 0.07645 lb/ft^3
- Viscosity: 1.202 X 10^{-5} lbm/ft-s
- Thermal Conductivity: 0.01399 btu/h-ft-°R
- Specific Heat: 0.2404 btu/lb-°R

The duct study, fin study, and manifold study were conducted assuming ideal initial conditions if these were dependent on another study. For example, the fin spacing simulation was dependent on the velocity of the inlet air. To perform the study, the inlet air was assumed to be perfectly even with a velocity of 25 mph. Similarly, the initial conditions of the duct and manifold were assumed to be ideal or at maximum load.

Design Point: 3 Air Duct Designs

The goal of the air duct is to bring cool air to the heat exchanger that is both offset by 18 in and 60 in downstream with maximum velocity and minimum total pressure drop. Inlet dimension is 12 in x 20 in, and area of outlet face is 300 in^2. Three designs were tested to meet these goals. All three designs use a constant outlet dimension of 15 in x 20 in.

Design 1: Direct Duct with Gradual Expansion

Geometry
The first design of the duct was a rectangular channel with an inlet face of 12 in x 20 in that expands to an outlet face of 15 in x 20 in. Figure 1shows the fluid region inside of the duct.

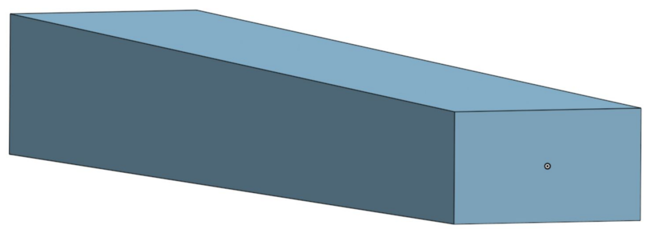


Figure 2.1. The fluid region inside duct design 1

Mesh
Multizone method is used to generate mesh, together with inflation on all boundary surfaces. Body sizing is used, with element sizing of 0.5 in, and inflation is set with 5 layers, and 1.2 growth rate. Named selection is created for inlet and outlet to be vinlet and poutlet respectively.

The same mesher parameters are used for design 2 and 3 as well.

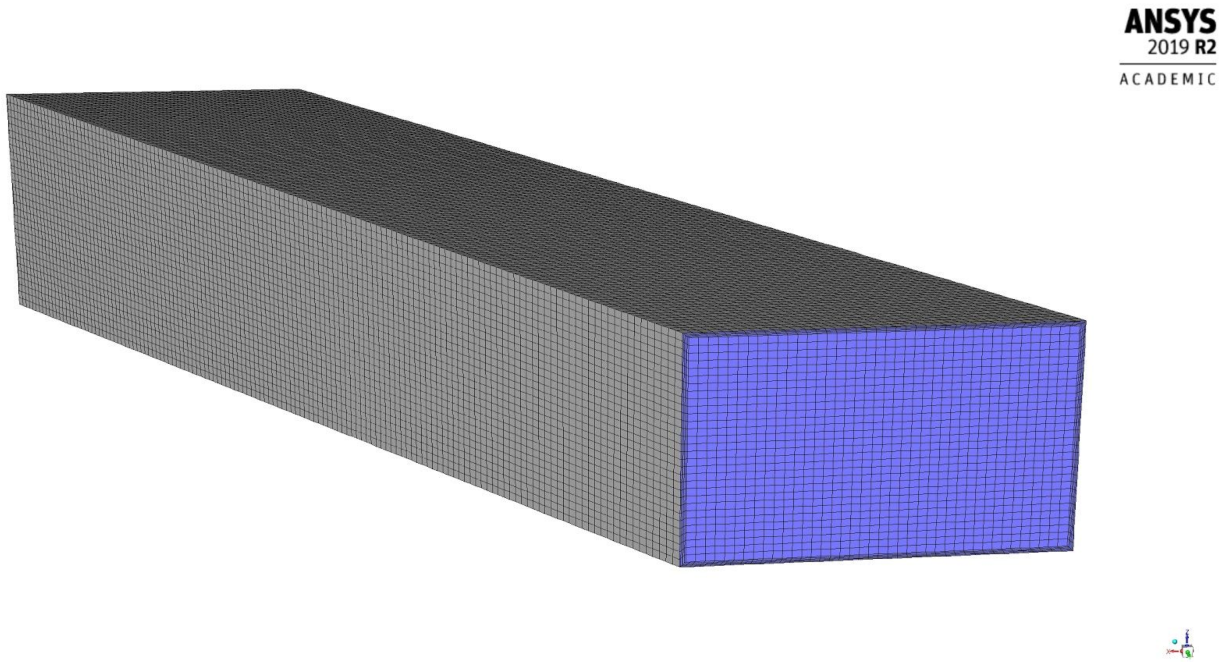


Figure 2.2. Mesh of fluid region design 1

Boundary Conditions

In fluent, a realizable K-E viscous model with all default parameters is turned on, and rest remains off. Inlet velocity is specified choosing “Magnitude, Normal to Boundary” method, and value is set at 2200 ft/min, or 25 mph.Turbulence parameters remain default, which is shown in table 1.1. The model is then solved using the default pressure-velocity coupled solver, with default parameters included in Table 1.2. All setup parameters in table 1.1 and table 1.2 apply to design 2 and 3 simulations as well.

Boundary Condition	Detail
Turbulence Model	realizable k-epsilon FLUENT default
Air Inlet Velocity	2200 ft/min

Table 3.1. Boundary conditions of duct simulation (design 1,2,3).

The following table outlines the solver parameters used for the single fin simulation.

Property	**Detail**
Scheme	Coupled
Gradient	Least Squares Cells Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Turbulent Dissipation Rate	First Order Upwind
Energy	FirstOrder Upwind
Pseudo Transient	On

Table 3.2. Solver settings of duct simulations (design 1,2,3).

Preliminary Hand Calculations

Continuity equations:

$$V_{airin} \times A_{in} = V_{airout} \times A_{out} \rightarrow V_{airout} = \frac{V_{airin} \times A_{in}}{A_{out}}$$

Plug in following parameters:

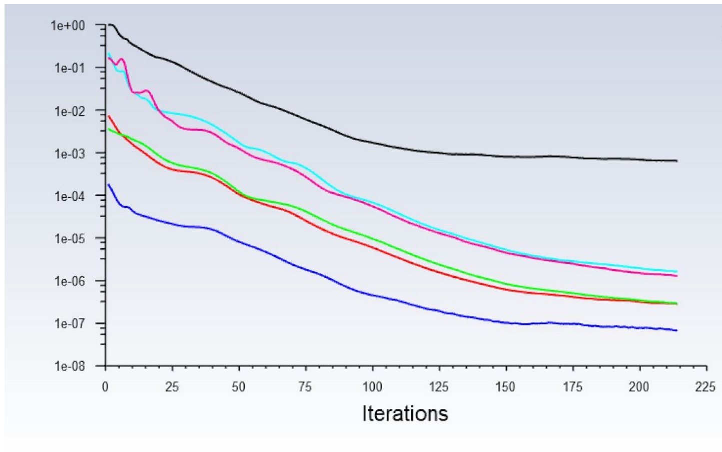
$A_{airin} = 12 \times 20in^2 = 1.667ft^2, A_{airout} = 2.083ft^2, V_{airin} = 2200ft/min$

Resultant outlet air velocity is:

$V_{airout} = 1760ft/min$

Simulation Results

The model converged after 200 iterations. Figure 4 shows that residuals drop at least 2 order of magnitudes, and average outlet velocity remains constant. The average outlet velocity is 1988 ft/min, and total pressure loss is 0.026 psi.



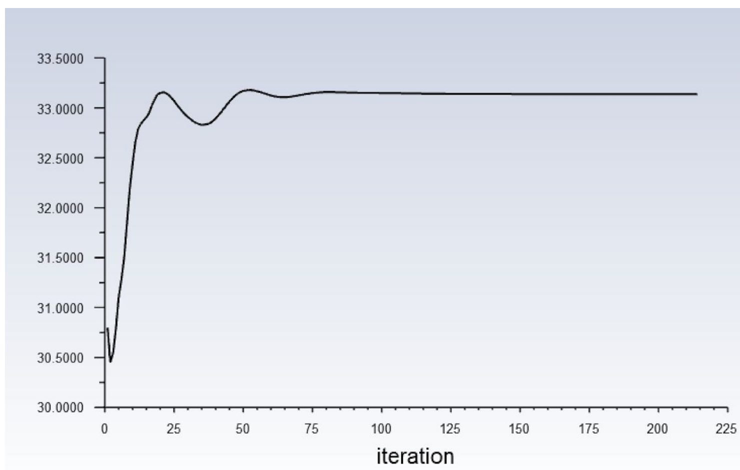


Figure 2.3. (Top) Residuals for the simulation of duct design 1. (Bottom) Average velocity of outlet face

Design 2: Direct Duct with Sudden Expansion

Geometry

The second design has the same 12 in x 20 in inlet and 15 in x 20 in outlet face as design 1. However, the duct undergoes a rapid expansion for the first 5 in, where surface area changes from 12x20 to 15x20, and the rest of downstream duct remains the same 15x20 cross area dimensions. Figure 5 shows a model of fluid inside duct design 2 with its inlet facing the page.

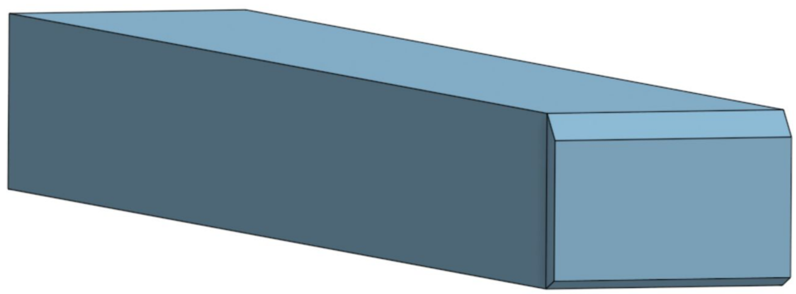


Figure 3.1. The fluid region inside duct design 2

Mesh and fluent set up

Design 2 uses identical mesher settings as design 1. The mesh is very similar to that of design 1. Design 2 also uses identical fluent boundary conditions and fluent solver setting as design 1. Thus, the settings are not reproduced for simplicity.

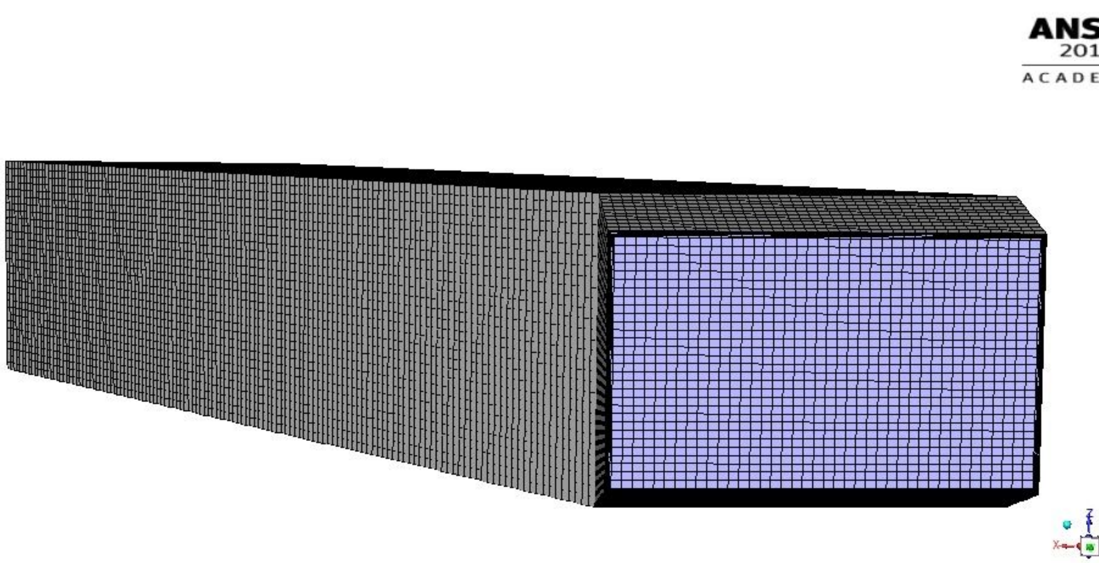


Figure 3.2. Mesh of fluid region design 2

Simulation Results

Design 2 model converged after about 120 iterations. In figure 2.3, residuals dropped at least 2 orders of magnitude, and average velocity on the outlet face kept constant. The result average velocity is 2058 ft/min, and total pressure loss is 0.012 psi.

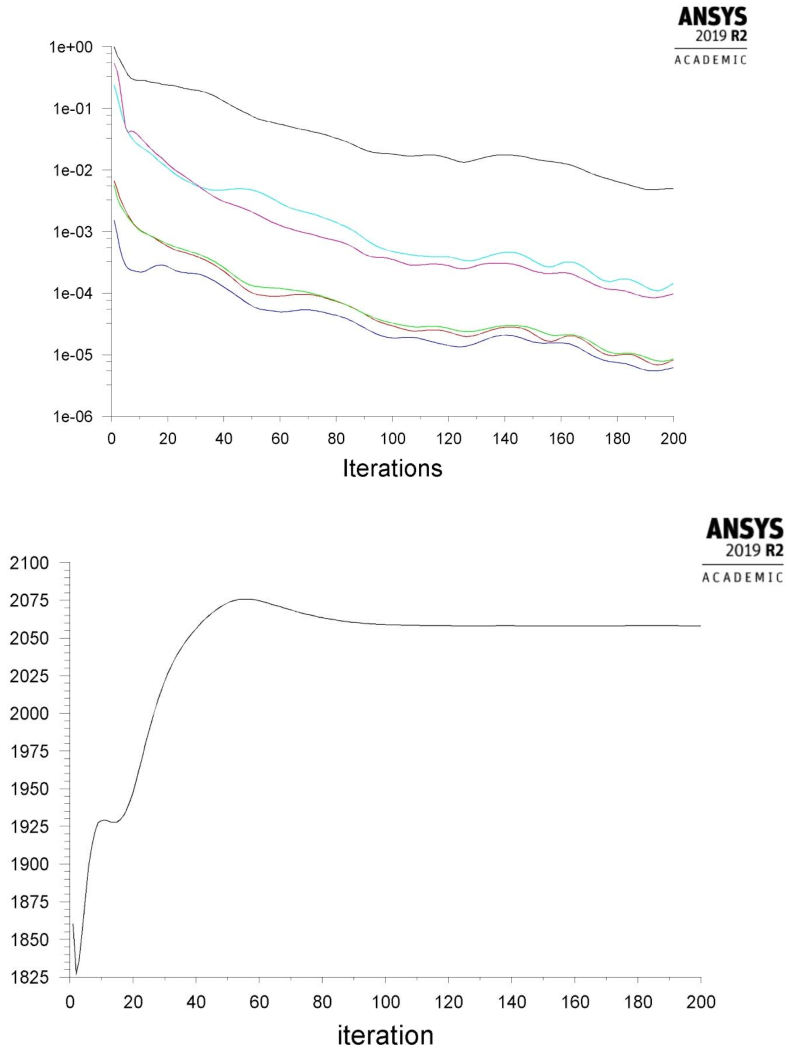


Figure 3.3. (Top) Residuals of duct design 2 simulation (Bottom) Average velocity of outlet face in ft/min

Design 3: Curved Duct with Sudden Expansion

Geometry

The third duct design has the same 12 in x 20 in inlet that expands to an 15 in x 20 in within the first 5 in of duct. The duct then extrudes to the outlet through a splined path with constant cross sections.

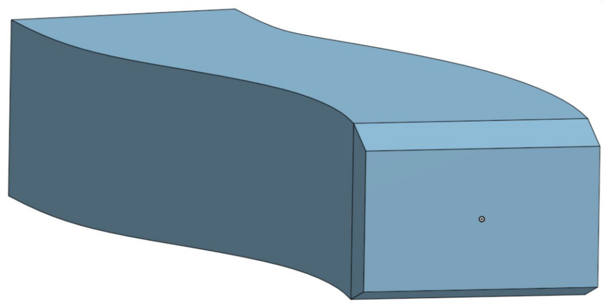


Figure 4.1. The fluid region inside duct design 3.

Mesh and fluent setup

Design 3 uses identical mesher settings as design 1. The mesh is very similar to that of design 1. Design 3 also uses identical fluent boundary conditions and fluent solver setting as design 1. Thus, the settings are not reproduced for simplicity.

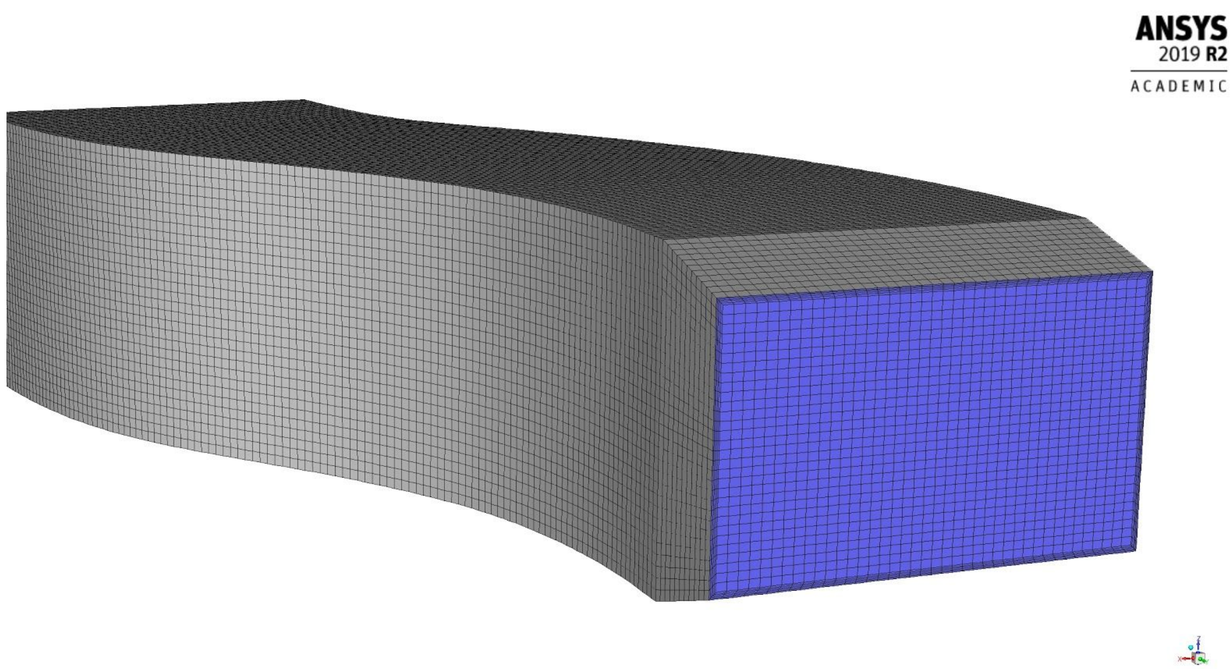


Figure 4.2. Mesh of fluid region design 3

Simulation Results

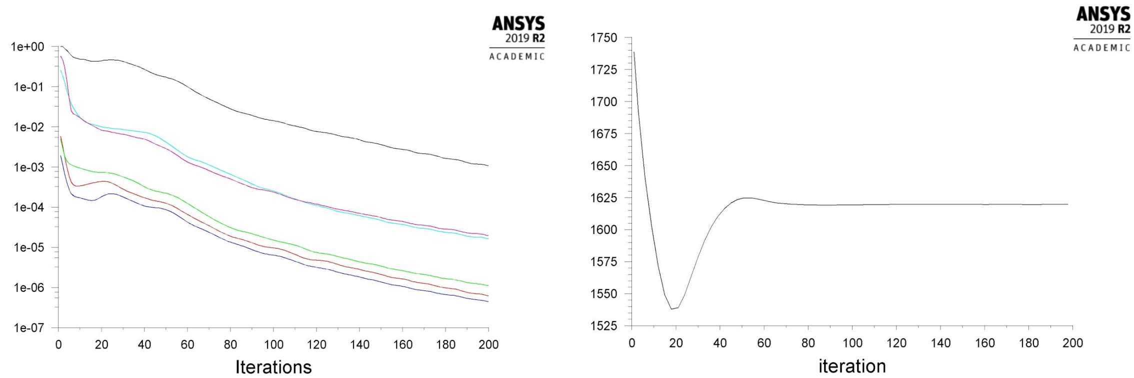


Figure 4.3. (Left) Residuals of duct design 3 simulation
(Right) Average velocity of outlet face in ft/min

Design 3 model converged after about 60 iterations. In figure 3.3, residuals dropped at least 2 orders of magnitude, and average velocity on the outlet face kept constant. The result average velocity is 1619 ft/min, and total pressure loss is 0.0074 psi.

Air Duct Design Conclusion

All three simulations behave in line with hand calculations. Average velocity loss and total pressure loss are listed in the following table and compared. It is clear that design 2 yields the least velocity decrease and a reasonable loss in total pressure. Thus, design 2 best fits the design criteria.

Design	Average Outlet Velocity (ft/min)	Average Outlet Total Pressure (psi)
1	1988	0.026
2	2058	0.012
3	1619	0.0074

Table 4. Result of Duct design study

Design Point: Manifold Design

Purpose

The goal of the manifold model was to identify an ideal manifold design to evenly distribute oil flow to all tubes/cores of the heat exchanger. A typical inlet oil manifold in a car radiator is fed in vertically from above and the oil is distributed to the tubes which flow horizontally. To ensure that velocities through all the tubes are equal, sometimes the manifolds include a taper angle in an attempt to maintain a similar static pressure upstream of all tubes. A parametric study was conducted in Fluent to assess what taper angle is necessary to achieve good flow distribution.

Geometry and Mesh

The manifold was analyzed as a 2D model for simplicity and was deemed good enough to compare flow distribution effects from different taper angles, inlet flow velocities, and etc. The geometry was created in DesignModeler so that dimensions could be set as parameters to be varied. The geometry is shown in Figure 5.1, depicts a long 0.5” height by 1.0” width by 20” length manifold with 20 rectangular tube outlets, each 0.1” height by 1.0” width (into the page).

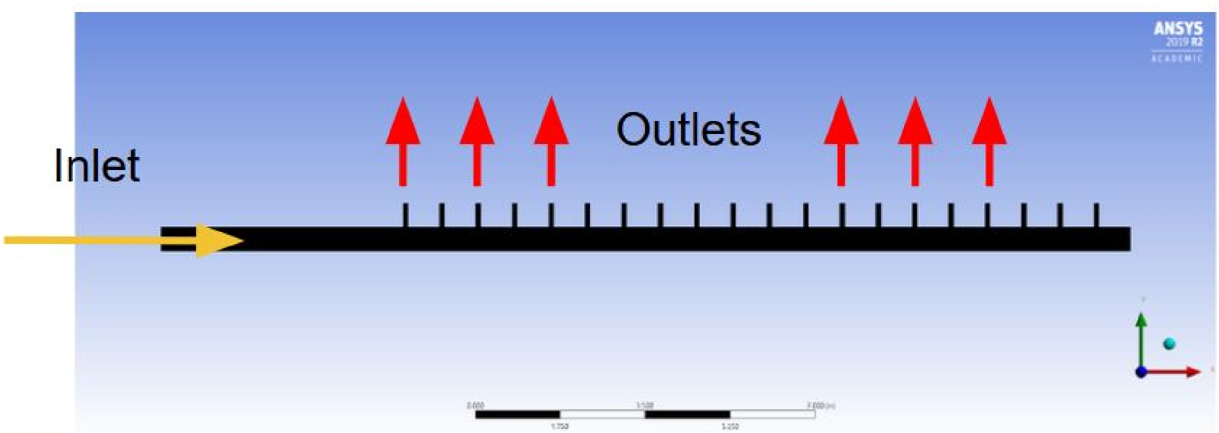


Figure 5.1. 2D Manifold geometry showing inlet and outlets

In Figure 5.1, a close-up of the mesh for the 2D manifold is shown. There are a total of **440,000 mesh elements** and with the range of velocities analyzed, the y+ value for the near wall elements is less than or equal to 1. Most of the elements are perfect quadrilaterals but with the introduction of a taper angle, there is some skewness but often negligible.

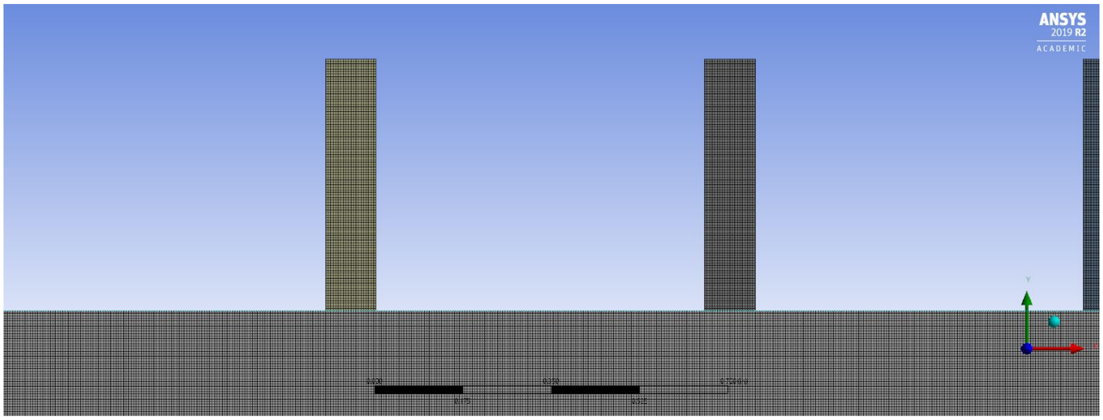


Figure 5.2. Close-up of mesh for 2D manifold

Boundary Conditions & Solver Setting

The boundary conditions are detailed in the table below. The material properties of the oil can be found under *Design Assumptions* on page 5.

Boundary Condition	**Detail**
Turbulence Model	K-epsilon, Realizable
Energy	Off
Oil Inlet Velocity	Variable

Table 5.1. The boundary conditions of the manifold model.

The following table outlines the solver parameters used for the 2D manifold simulations.

Property	**Detail**
Scheme	Coupled
Gradient	Least Squares Cells Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Turbulent Dissipation Rate	Second Order Upwind

Table 5.2. The solver settings used for the manifold model.

Simulation Results & Calculations

Using the coupled solver and running in parallel, the simulations were set for 1000 iterations. The solutions typically converged in < 500 iterations.

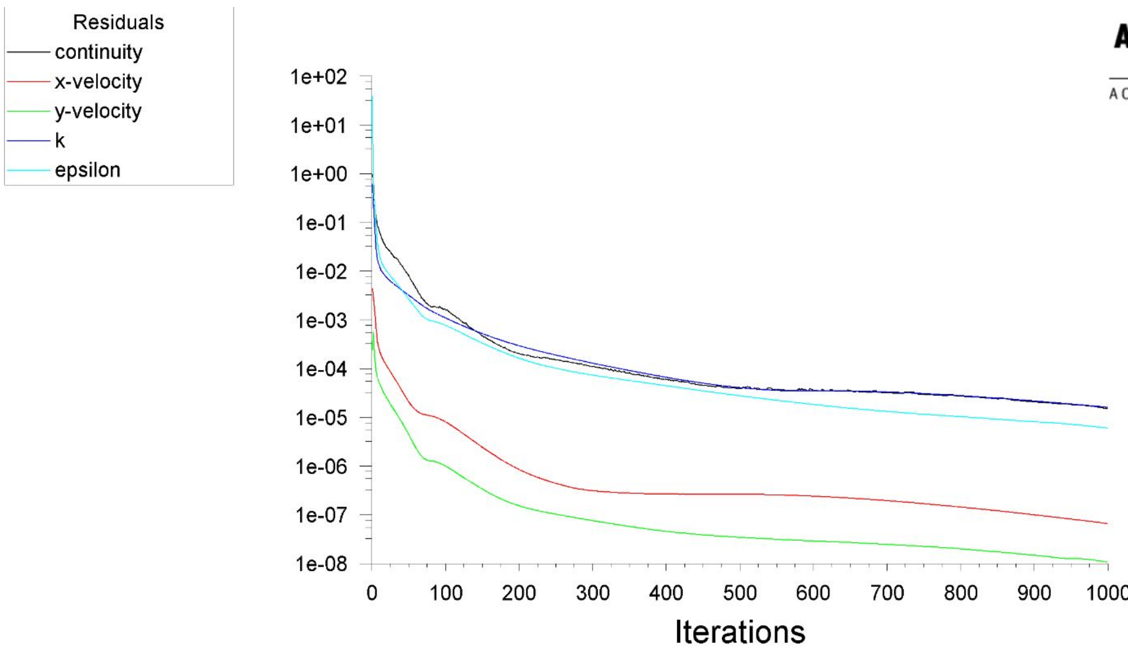


Figure 5.3. Typical residuals for 2D manifold simulations.

Based on initial $\dot{m} = \rho v A$ calculations and using 4.0 -5.5 gpm of oil from the requirements, **400 ft/min** of oil was chosen as the **inlet velocity**. The total area of the outlets is ~4x the total area of the inlet, so the **exit velocity** out of each tube/outlet should ideally be **~100 ft/min**.

Figure 5.4 shows the velocity and static pressure contours of a non-tapered manifold. Based on the velocity contour, it is observable that core 20 has a higher velocity than core 1. The static pressure contour also shows that the pressure varies significantly along the length of the manifold, leading to uneven flow distribution.

The "end diameter" of the manifold was parameterized to understand how adding a taper angle to the manifold would affect the flow distribution to the cores. A smaller end diameter indicates a larger taper.

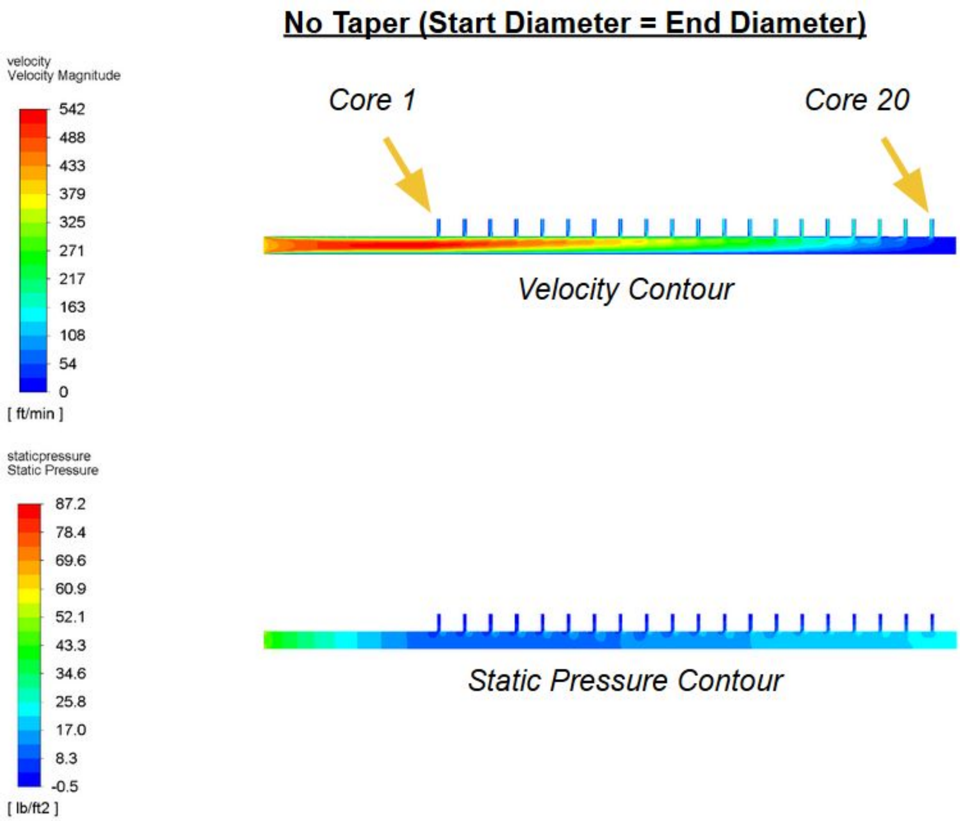


Figure 5.4. Velocity and static pressure contour plots of non-tapered manifold.

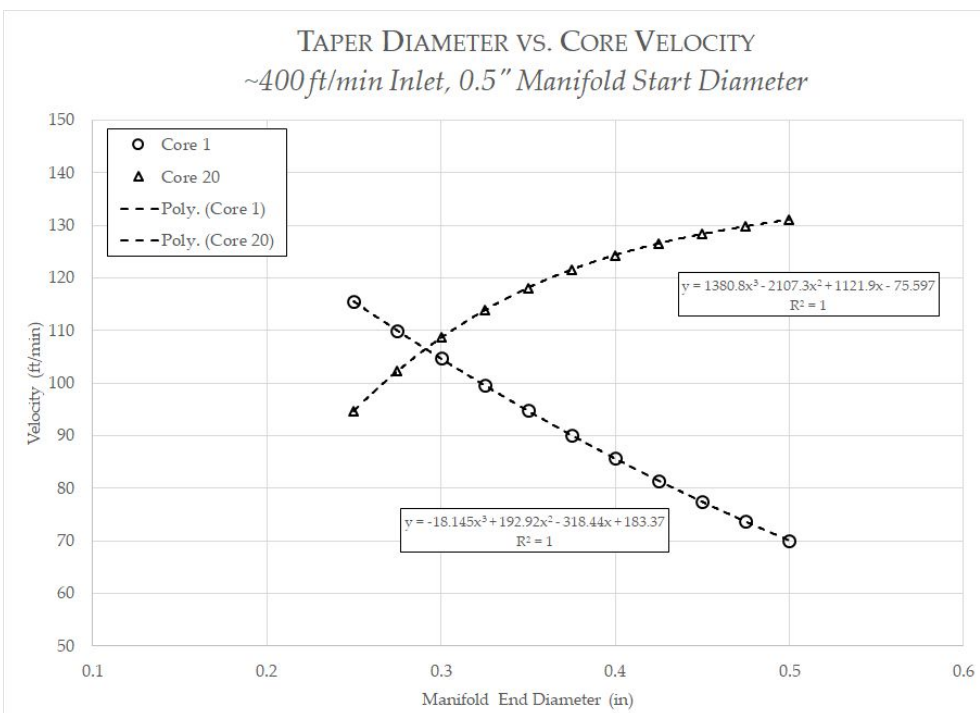


Figure 5.5. Velocities of Core 1 and Core 20 at different manifold end diameters.

From the parametric study of the manifold diameter, **core 1 and core 20 facet average exit velocities were recorded**. The results are plotted in Figure 5.5. With no taper of the manifold (end diameter = 0.5”), there is significant discrepancy in the velocities between the first and last core: ~70 ft/min vs. ~130 ft/min. By including the taper angle, the flow distribution in the manifold improves and reaches an **optimum around 0.29” end diameter**.

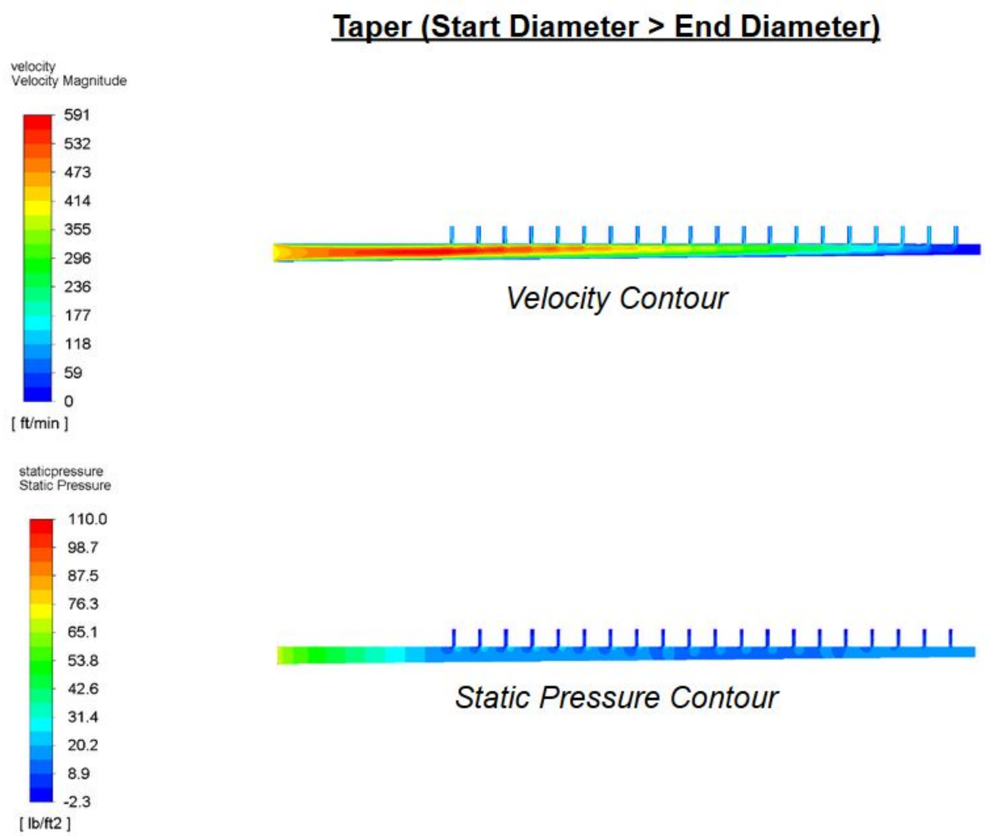


Figure 5.6. Velocity and static pressure contours of tapered manifold (with 0.30” end diameter).

Contours of the 0.30” end diameter simulation is plotted above in Figure 5.6. The velocity contour shows that the velocities are at least more even in color throughout all cores. The static pressure contour shows significant improvement in uniformity compared to the contour from the non-tapered manifold.

Conclusion

By incorporating the 0.30” end diameter taper, there is a uniform distribution of velocities from core to core, which is critical to the predictability as well as efficiency of the heat exchanger**.** Due to time constraints, the effects of Reynold’s number (which would be mainly changes in velocity) on the taper angle was not fully characterized. Initial studies suggest that there are some effects to the uniformity, but not substantial.

Design Point: Fin Thickness

Purpose

The goal of this design study was to understand how fin thickness affects the heat transfer of the oil to the air. Another goal of this study is to assess how oil velocity affects heat transfer after choosing the fin thickness.

Geometry & Mesh

The single fin model consisted of an oil fluid zone, air zone, and fin modeled using ANSYS Design Modeler. The thickness of the fin is 0 in Design Modeler and is simulated as a thin wall with a variable thickness in ANSYS Fluent. The geometry of the single fin model is described in the table below.

Part	X Width (in)	Y Height (in)	Z Length (in)
Oil Zone	1.00	0.10	0.50
Air Zone	4.00	0.50	0.50
Fin	2.00	0.50	VARIABLE

Table 6.1. The width, height, and length of the single fin model.

The figures below show the air fluid zone in blue, oil zone, and location of the thin fin.

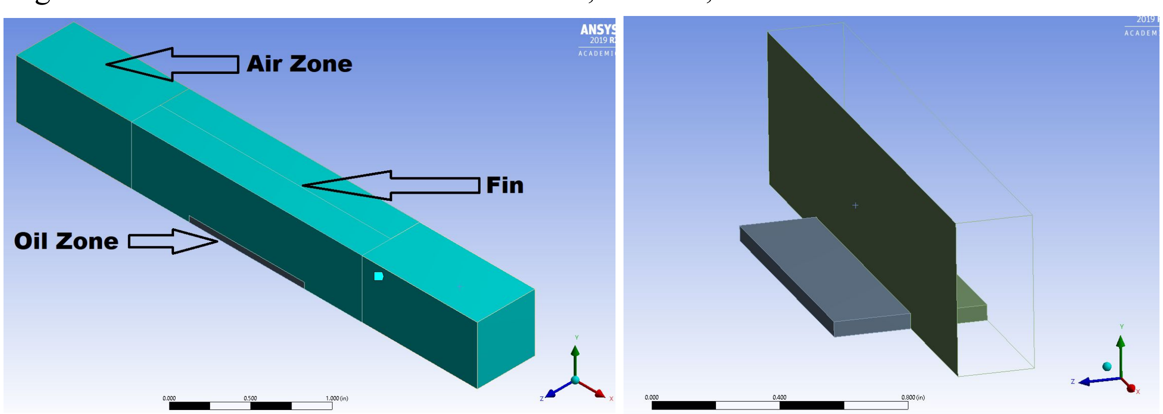


Figure 6.1. The air fluid zone in blue, oil fluid zone, and thin fin (Left). Only the oil fluid zone and fin (Right)

The parameters used for meshing are detailed in the table below.

Mesh Parameter	**Detail**
Overall Method	Multizone, Hexa/Prism
Air Inlet/Outlet Zone Body Sizing	0.040 in
Middle Air Zone Body Sizing	0.025 in
Oil Zone Body Sizing	0.007 in
Inflation around Oil Tube Boundary	8 layers with 0.0025 in first layer and 1.2 growth rate

Table 6.2. The parameters used to mesh the single fin model.

The figures following show the single fin model meshed. The first image is a front view of the entire model to show the varying sizes of each region. The second image is a zoomed view of the oil zone boundary to highlight the inflation. The final image is an isometric view of the transition region between the outlet air zone and the middle air zone to emphasize the mesh size adaptation.

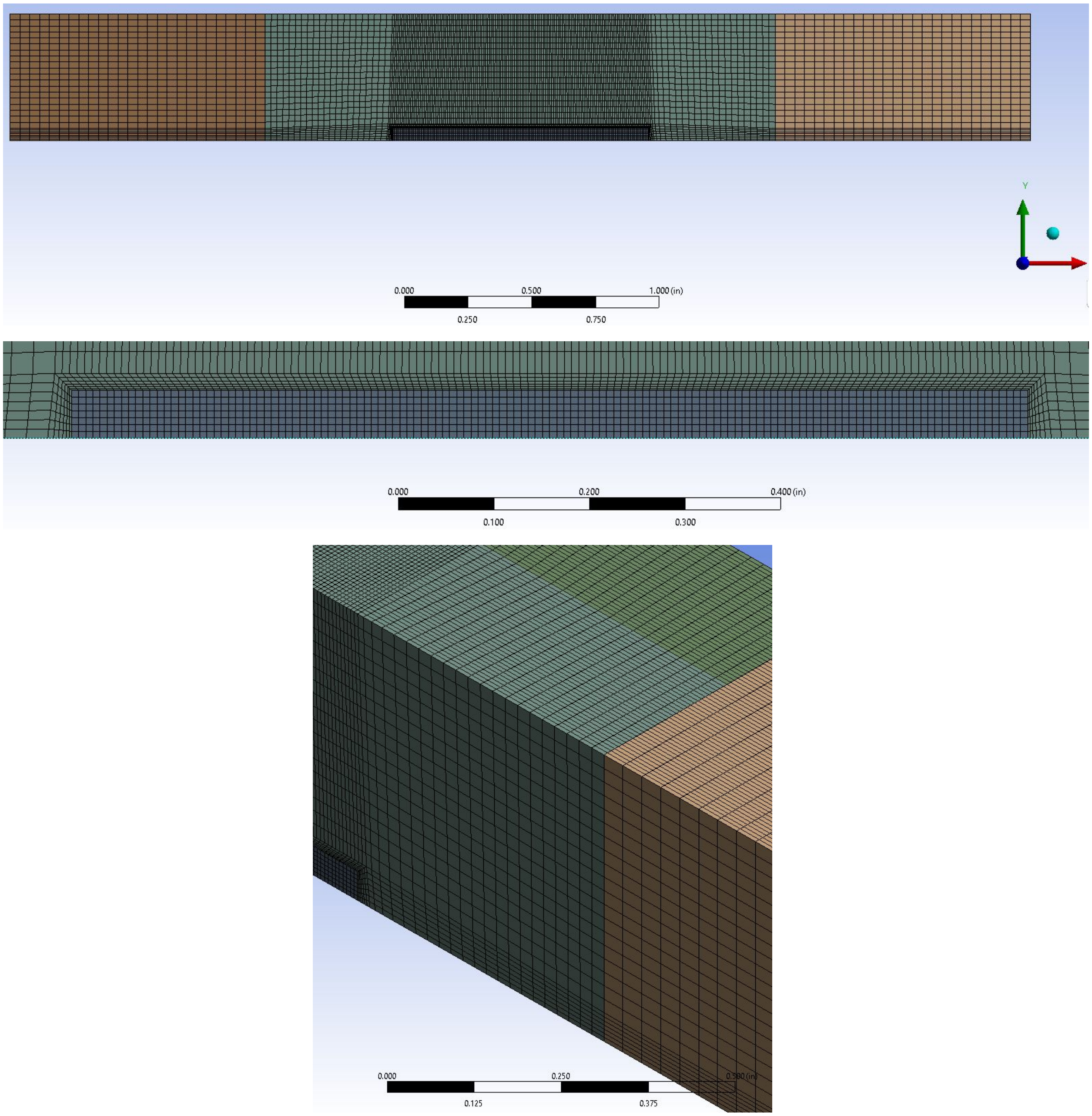


Figure 6.2. A front view of the overall mesh (Top). A zoomed in view of the oil boundary inflation (Middle). An isometric view of the transition region between the middle and outlet air zone (bottom).

Boundary Conditions & Solver Setting

The boundary conditions are detailed in the table below. The material properties of the air and oil can be found under *Design Assumptions* on page 5. There were two sets of simulations done using the geometry and mesh as defined previously. First, the fin thickness is varied while keeping the oil inlet velocity constant at 50 ft/min. Then, the ideal fin thickness, which is found to be 0.015 in, is kept constant while varying the oil velocity to assess how many cores would be needed in the heat exchanger to reach the desired oil exit temperature. Finally, once a oil velocity was chosen, the fin thickness was varied again to ensure that oil velocity does not substantially affect the optimum fin thickness.

Boundary Condition	Detail
Turbulence Model	realizable k-epsilon FLUENT default
Air Inlet Velocity	2000 ft/min
Oil Inlet Velocity	50 ft/min or VARIABLE
Air Inlet Temperature	108°F
Oil Inlet Temperature	350°F
Fin Thickness	0.015 in or VARIABLE

Table 6.3. The boundary conditions of the single fin model.

The following table outlines the solver parameters used for the single fin simulation.

Property	**Detail**
Scheme	Coupled
Gradient	Least Squares Cells Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Turbulent Dissipation Rate	Second Order Upwind
Energy	Second Order Upwind
Pseudo Transient	On

Table 6.4. The solver settings used for the single fin model.

Simulation Results & Calculation Verification

The simulation was run for a total of 300 iterations because the calculation seemed to converge at around 250 iterations. The following graphs show the residual graph for the simulation and the area weighted average temperature of the oil exiting the single fin model.

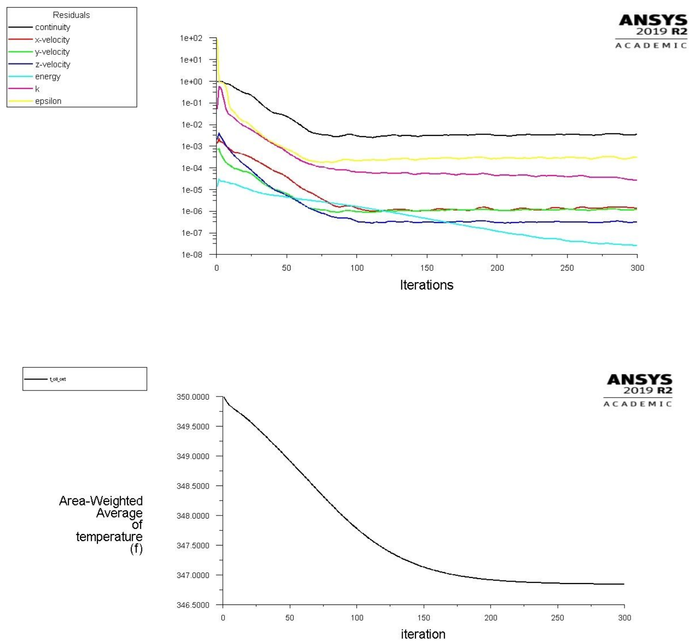


Figure 6.3. The residual graph of the single fin simulation and the area weighted average outlet temperature of the oil.

The table below shows all the design points in the parametric study, varying fin thickness and oil velocity. The design points are split into 3 main sections, each with its own design goal. **Ultimately, a fin thickness of 0.015” and an oil velocity of ~20 ft/min (which corresponds to 40 tubes sized to 0.1” x 1” cross section) was chosen.**

Name	Fin Thickness (in)	Oil Velocity (ft/min)	Exit Air Temp (F)	Exit Oil Temp (F)	Design Goal
DP 0	0.005	50.0	121.89	347.00	Maintain oil velocity and vary fin thickness
DP 1	0.0075	50.0	122.76	346.89	
DP 2	0.01	50.0	123.32	346.69	
DP 3	**0.015**	**50.0**	**124.01**	**346.66**	
DP 4	0.02	50.0	124.43	346.64	
DP 5	0.04	50.0	125.17	346.46	
DP 6	0.015	39.4	123.94	345.87	Maintain fin thickness and vary oil velocity
DP 7	0.015	29.5	123.81	344.72	
DP 8	0.015	24.6	123.72	343.83	
DP 9	**0.015**	**19.7**	**123.60**	**342.57**	
DP 10	0.015	9.8	123.10	337.04	
DP 11	0.02	19.7	124.00	342.49	Choose 0.1 m/s oil velocity and double check that changing fin thickness does not change result
DP 12	0.03	19.7	124.45	342.51	
DP 13	0.04	19.7	124.72	342.65	

Table 6.5. Parametric study results for the single fin model

The following contours show the temperature profiles of various parts of the single fin model.

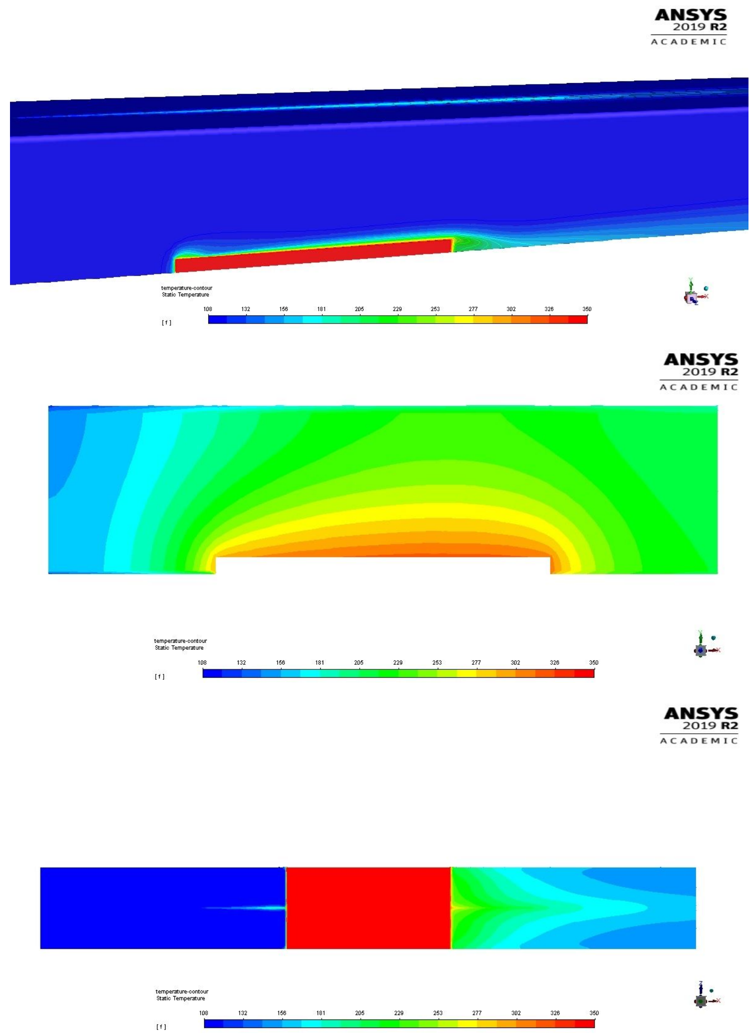


Figure 6.4. Temperature Contours: Isometric view of the oil inlet with air coming in from the left (Top). Front view of the top half of the fin with air coming in from the left (Center). Bottom view of the model, looking at the plane of symmetry with air coming in from the left (Bottom). The temperature scales from 108°F to 350°F.

The bulk of the air at the walls remain at 108°F as it passes over the single fin. The region around the tube carrying the oil becomes hot at around 229°F. The heat transfers to the downstream air. According to the temperature contour of the symmetry plane, the region directly aligned with the fin has the greatest increase in temperature. This is expected because of the heat transfer from the fin to the surrounding air. A closer look at the fin shows that the incoming air at 108°F cools the front of the fin to about 156°F while and the end of the fin to about 253°F, which is about a 100°F temperature gradient across the fin.

The energy between the air and oil is confirmed using a hand calculation.

$$\dot{Q}_{oil} = \dot{m}c_{oil}\Delta T_{oil} \text{ where } \dot{m} = \rho\dot{V} = \rho\nu A$$

$$A \equiv \text{tube cross sectional area}.\ \nu \text{ velocity}.\ \rho \text{ density}.$$

$$\dot{Q}_{oil} = \rho_{oil}\nu_{oil}A_{tube}c_{oil}\Delta T_{oil}$$

$$\dot{Q}_{oil} = (53.68\tfrac{lbs}{ft^3})(19.7\ \tfrac{ft}{min})(0.000347\,ft^2)(0.5\ \tfrac{btu}{lbm*°R})(807.96°R - 809.67°R)$$

$$\dot{Q}_{oil} = -1.36\tfrac{btu}{min}$$

$$\dot{Q}_{air} = \dot{m}c_{air}\Delta T_{air} \text{ where } \dot{m} = \rho\dot{V} = \rho\nu A$$

$$A \equiv \text{tube cross sectional area}.\ \nu \text{ velocity}.\ \rho \text{ density}.$$

$$\dot{Q}_{air} = \rho_{air}\nu_{air}A_{duct}c_{air}\Delta T_{air}$$

$$\dot{Q}_{air} = (0.07645\tfrac{lbs}{ft^3})(2000\ \tfrac{ft}{min})(0.00174\,ft^2)(0.2404\tfrac{btu}{lbm*°R})(582.91°R - 567.67°R)$$

$$\dot{Q}_{air} = 0.996\ \tfrac{btu}{min}$$

There is a 15.5% difference between the two values, which is a large difference. It is unclear why such a difference exists, but it is likely that the average air temperature reported by the simulation is an undervalued average.

Conclusion

Based on a parametric study of a single-fin model, results suggest 0.015" thick fin with 0.1 m/s oil velocity (which corresponds to 40 tubes) would be a decent selection. The temperature contour produced expected temperature ranges with the expected heat spread. The energy of the model was not accurately conserved. The undervalued average outlet air temperature is likely the cause of the energy imbalance. The average outlet air temperature is expected to be higher; but this does not change the result of the oil temperature.

Design Point: Fin Spacing

Purpose

The goal of the three fin model was to find the ideal spacing as to balance the amount of heat able to be transferred which decreases with fin spacing and the total number of fins that can be squeezed into the same space. The fin spacing is varied from 0.25 in to 0.075 in, varying by 0.025 in to have a wide fin spacing range with many incremental points.

Geometry & Mesh

The three fin model consisted of an oil fluid zone, air zone, and three fins modeled using ANSYS Design Modeler. The thickness of the fin is 0 in Design Modeler and is simulated as a thin wall with a variable thickness in ANSYS Fluent. The geometry of the three fin model is described in the table below.

Part	X Width (in)	Y Height (in)	Z Length (in)
Oil Zone	1.00	0.10	0.50
Air Zone	4.00	0.50	0.50
Fin	2.00	0.50	0.00*
Fin Spacing	0.00	0.00	VARIABLE

Table 7.1. The geometry of the various parts of the three fin model according to XYZ length. *The thickness of the fin is 0 in Ansys Design Modeler because it is treated as a thin wall and is given a thickness in ANSYS Fluent of 0.015 in.

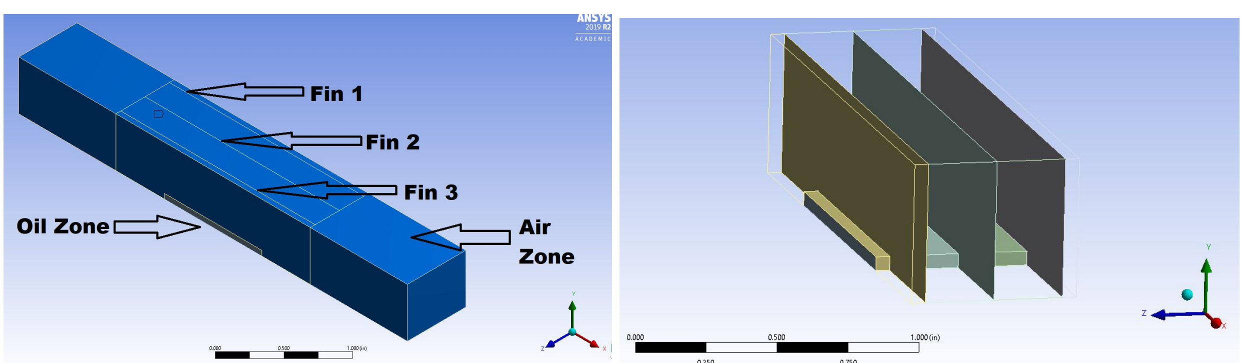


Figure 7.1. The air fluid zone in blue, oil fluid zone, and thin fin (Left). Only the oil fluid zone and fin (Right)

The parameters used for meshing are detailed in the table below.

Mesh Parameter	Detail
Overall Method	Multizone, Hexa/Prism
Air Inlet/Outlet Zone Body Sizing	0.040 in
Middle Air Zone Body Sizing	0.025 in
Oil Zone Body Sizing	0.010 in
Inflation around Oil Tube Boundary	8 layers with 0.0015 in first layer and 1.2 growth rate

Table 7.2. The mesh parameters for the three fin model

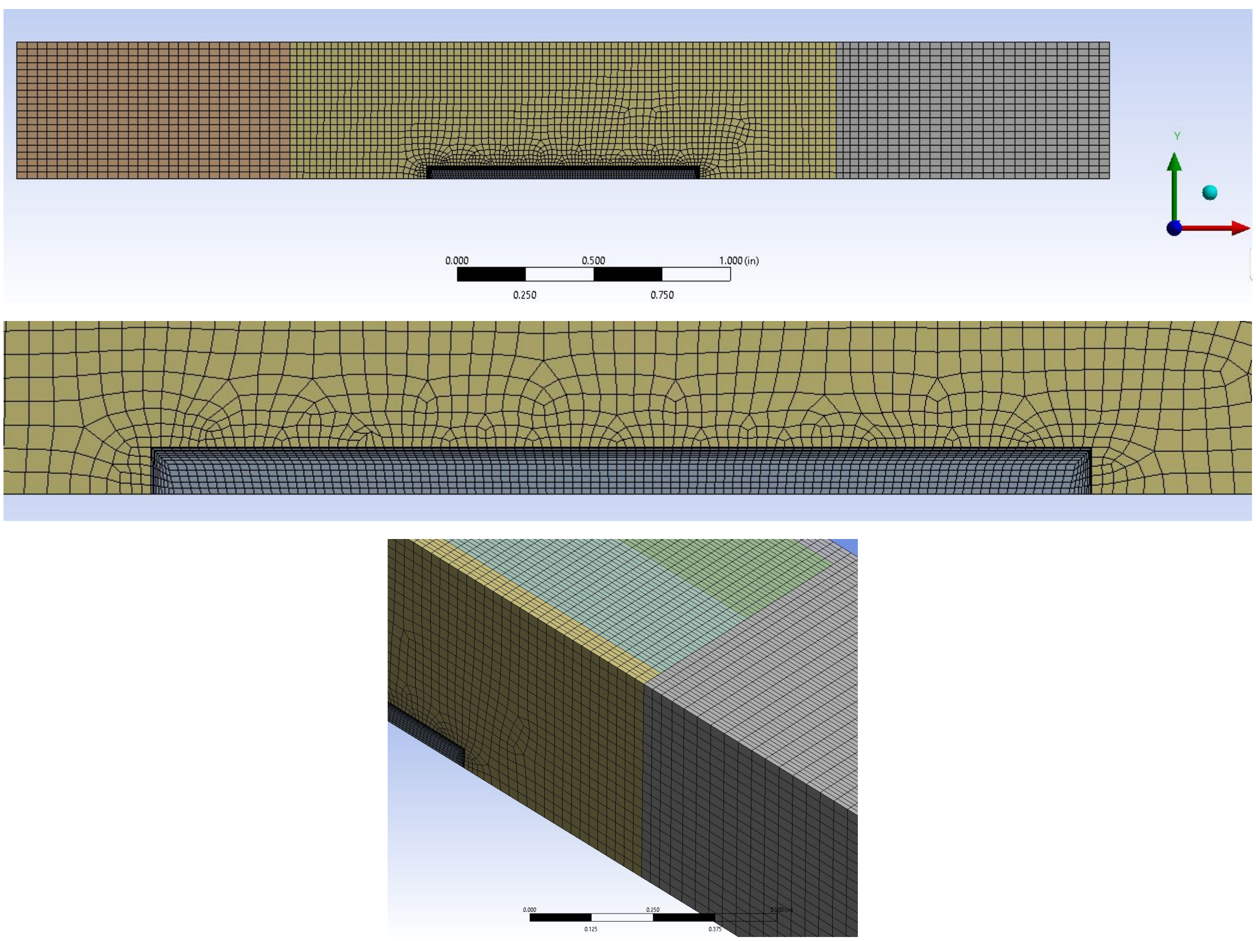


Figure 7.2. A front view of the overall mesh (Top). A zoomed in view of the oil boundary inflation (Middle). An isometric view of the transition region between the middle and outlet air zone (bottom).

Boundary Conditions & Solver Setting

The boundary conditions are detailed in the table below. The material properties of the air and oil can be found under *Design Assumptions* on page 5. There were two sets of simulations done using the geometry and mesh as defined previously. The fin spacing was varied from 0.25 in to 0.075 in, varying by 0.025 in to have a wide fin spacing range with many incremental points.

Boundary Condition	Detail
Turbulence Model	realizable k-epsilon FLUENT default
Air Inlet Velocity	2000 ft/min
Oil Inlet Velocity	20 ft/min
Air Inlet Temperature	108°F
Oil Inlet Temperature	350°F
Fin Thickness	0.015 in

Table 7.3. The boundary conditions of the three fin model.

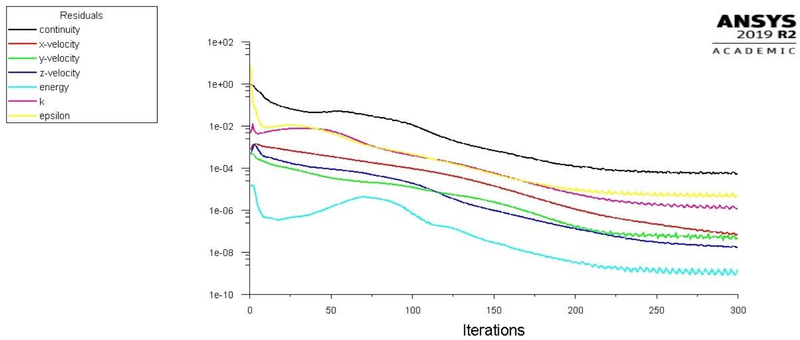
The following table outlines the solver parameters used for the single fin simulation.

Property	**Detail**
Scheme	Coupled
Gradient	Least Squares Cells Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Turbulent Dissipation Rate	Second Order Upwind
Energy	Second Order Upwind

Table 7.4. The solver settings used for the three fin model.

Simulation Results & Calculation Verification

The simulation was run for a total of 300 iterations because the calculation seemed to converge at around 150 iterations. The following graphs show the residual graph for the simulation and the area weighted average temperature of the oil exiting the threefin model.



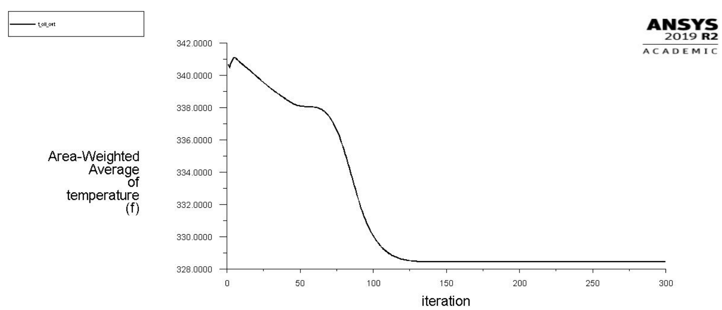


Figure 7.3. The residual graph of the single fin simulation and the area weighted average outlet temperature of the oil.

Fin Spacing (in)	Average Air Outlet Temperature (°F)	Average Oil Outlet Temperature (°F)
0.250	131.00	337.87
0.225	128.48	338.05
0.200	128.03	338.34
0.175	127.62	338.52
0.150	127.60	338.69
0.125	127.62	338.84
0.100	128.15	338.96

Table 7.5. Results of the parametric study on fin spacing on outlet air and oil temperatures

Based on the results from Table 7.5, a fin spacing greater than 0.150 in is when the oil outlet temperature remains significantly hotter. Any spacing below 0.150 in is an efficient spacing method.

Assuming a path length of 18 in and a fin thickness of 0.015 in:

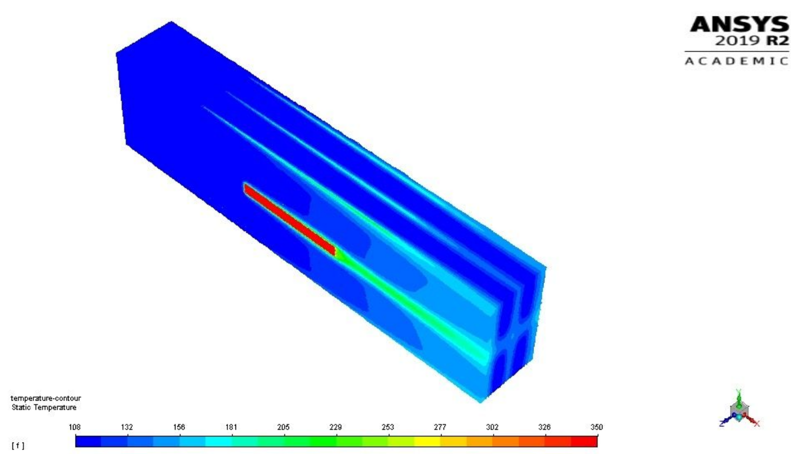
$$\#_{fin} = \frac{path\ length}{fin\ thickness + fin\ spacing}$$

Fin Spacing (in)	Maximum Number of Fins
0.250	72
0.225	80
0.200	90
0.175	102
0.150	120

Table 7.6. The total number of fins that can fit on the heat exchanger face

By using a spacing of 0.150 in, a total of 109 fins can fit; but it is unlikely that 109 fins are needed to drop the oil temperature down by the required amount. More details about the number of fins necessary to meet the cooling requirements can be found in *Data Verification via Hand Calculation*. Ultimately, a fin spacing of 0.25 in was chosen since it proved enough to cool the oil temperature below the target temperature of 195 deg F.

The following temperature contours show various parts of the three fin model at different viewing angles. All of the temperature scales range from 108°F to 350°F.



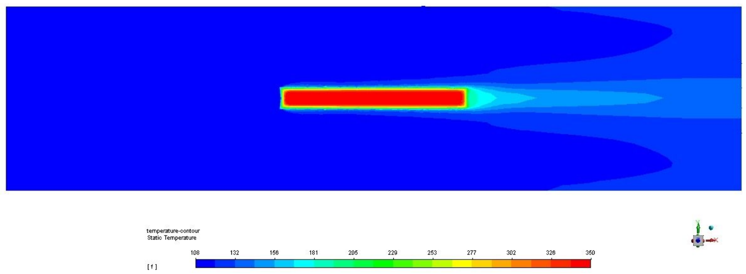


Figure 7.4. Temperature Contour with air coming in from the left: Isometric view of the three fin model with symmetric reflection to show the entire oil carrying tube passing through a section of air with three fins (Top). A front view of the oil inlet face (Center). The symmetry plane of the three fin model (Bottom).

Oil heats up fins, which then heats up air, indicated by a transition from blue to light green in the temperature contour graph. The outlet temperature profile of the air is not well spread. The front of the fin that first meets the is cooled to about the temperature of the inlet air at 108°F as indicated by the sharp light blue extrusions to the left of the oil carrying tube in the symmetry plane temperature contour. The back side of the fin is not cooled as much because the air has already gained the heat from the front of the fin. This is seen with the light blue to green region at the right side of the oil carrying tube in the symmetry plane temperature contour.

The following temperature contours show the temperature profile of the fin closest to the oil inlet and the oil carrying tube surface. The temperature profiles of all three fins were almost identical with the temperature surrounding the oil carrying tube slightly lower for the second and third fin. The temperature scale ranges from 108°F to 350°F.

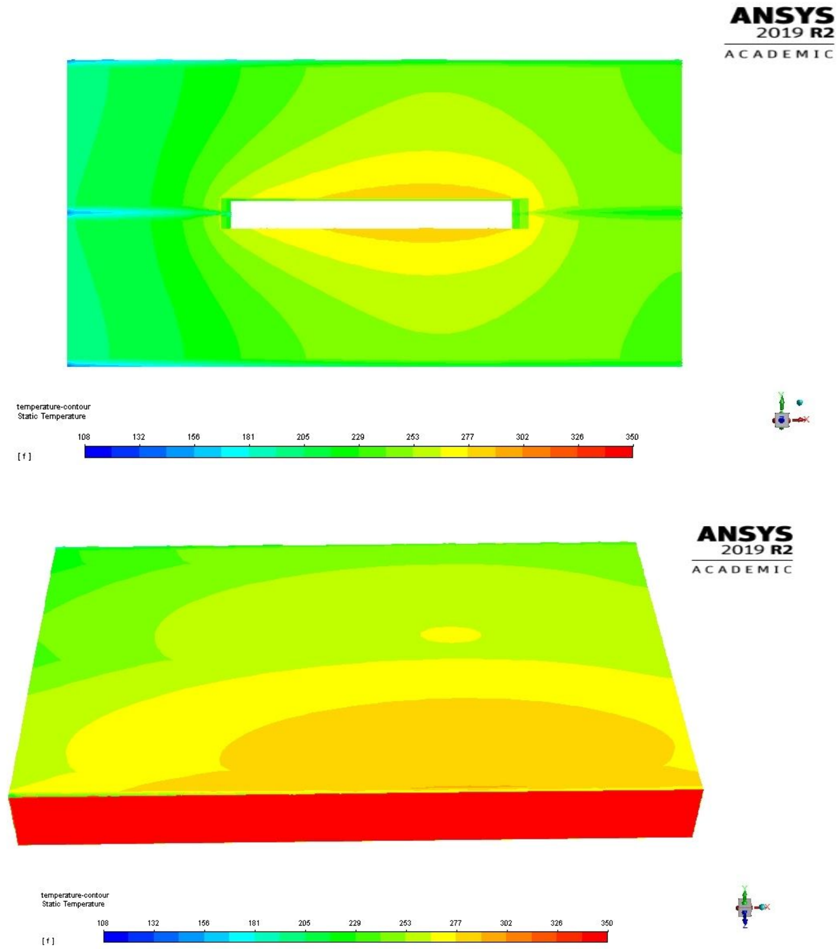


Figure 7.5. Temperature Contour: The front view of the first fin, the closest fin to the oil inlet (Top). An isometric view of the top of the oil carrying tube with the oil inlet (Bottom).

The oil temperature decreases as the oil moves through the 0.25 in tube as evidenced by the shift from dark yellow (~277°F) to bright green (~200°F) on the tube surface from inlet to outlet. The fin effectively transfers the heat to the surrounding air as shown by the fin temperature contour with an average temperature range of about 230°F, which is almost right between 108°F and 350°F. As expected, the front of the fin that first meets the incoming air is cooler than the back end of the fin.

The energy between the air and oil is confirmed using a hand calculation.

$\dot{Q}_{oil} = \dot{m}c_{oil}\Delta T_{oil}$ *where* $\dot{m} = \rho\dot{V} = \rho v A$

$A \equiv$ *tube cross sectional area* . v *velocity* . ρ *density* .

$\dot{Q}_{oil} = \rho_{oil}v_{oil}A_{tube}c_{oil}\Delta T_{oil}$

$\dot{Q}_{oil} = (53.68\frac{lbs}{ft^3})(50\ \frac{ft}{min})(0.000694\,ft^2)(0.5\ \frac{btu}{lbm*°R})(798.184°R - 809.67°R)$

$\dot{Q}_{oil} = \ -2.21\ \frac{btu}{min}$

$\dot{Q}_{air} = \dot{m}c_{air}\Delta T_{air}$ *where* $\dot{m} = \rho\dot{V} = \rho v A$

$A\ \equiv$ *tube cross sectional area* $.\ v$ *velocity* $.\ \rho$ *density* .

$\dot{Q}_{air} = \rho_{air}v_{air}A_{duct}c_{air}\Delta T_{air}$

$\dot{Q}_{air} = (0.07645\frac{lbs}{ft^3})(2000\ \frac{ft}{min})(0.00174\,ft^2)(0.2404\frac{btu}{lbm*°R})(592.50°R - 568.0°R)$

$\dot{Q}_{air} = \ 1.50\ \frac{btu}{min}$

There is a similar difference of 16.7% compared to the single fin study of 15.5%.
The difference is likely due to the average air temperature.

Fluid	Pressure Drop (psi)
Air	7.0e-3
Oil	2.7e-3

Table 7.7. Pressure drop across three-fin model for both air and oil

Pressure drop values were computed in FLUENT by taking the difference between the total pressure at the inlets and the total pressure at the outlets. The facet average pressure drops for the three-fin model are given in **Table 7.7**. These pressure drops will be multiplied accordingly to calculate the overall pressure drops of the fluids in the final heat exchanger design.

In addition to the simulations mentioned above, this model was solved again with updated boundary conditions (specifically inlet air temperature) and explained in further detail in *Overall Heat Exchanger Design*.

Conclusion

The temperature contours of the three fin model matched the expected temperature spread due to the hot fins and tube. Due to the fact that the energy balance was not kept in this calculation, the results must be looked at with some skepticism. By comparing the results of outlet oil temperature of the three fin study to the single fin study, there is an expected approximate 3 times the temperature drop when three fins are modeled. The air, on the other hand, does not increase temperature by a proportional amount. As long as the oil temperature drop is somewhat accurate, results of the three fin study can be extrapolated.

Overall Heat Exchanger Design

The IC engine heat exchanger will be constructed following a typical finned tube radiator design following the maximum size constraints of 300 in^2 with no dimension exceeding 36 in. The design was chosen to be 15 in X 20 in to maximize the area exposed to the incoming air. **Figure 8.1** shows a rendering of the proposed heat exchanger design. Aluminum was chosen due to its high corrosion resistance, low density, decent strength, ease of manufacturing, and it is the most common material used in the car radiator industry. It features 2 columns of 20 rows of tubes/cores for a total of 40 tubes/cores. Each column is equipped with its own inlet manifold and exit manifold.

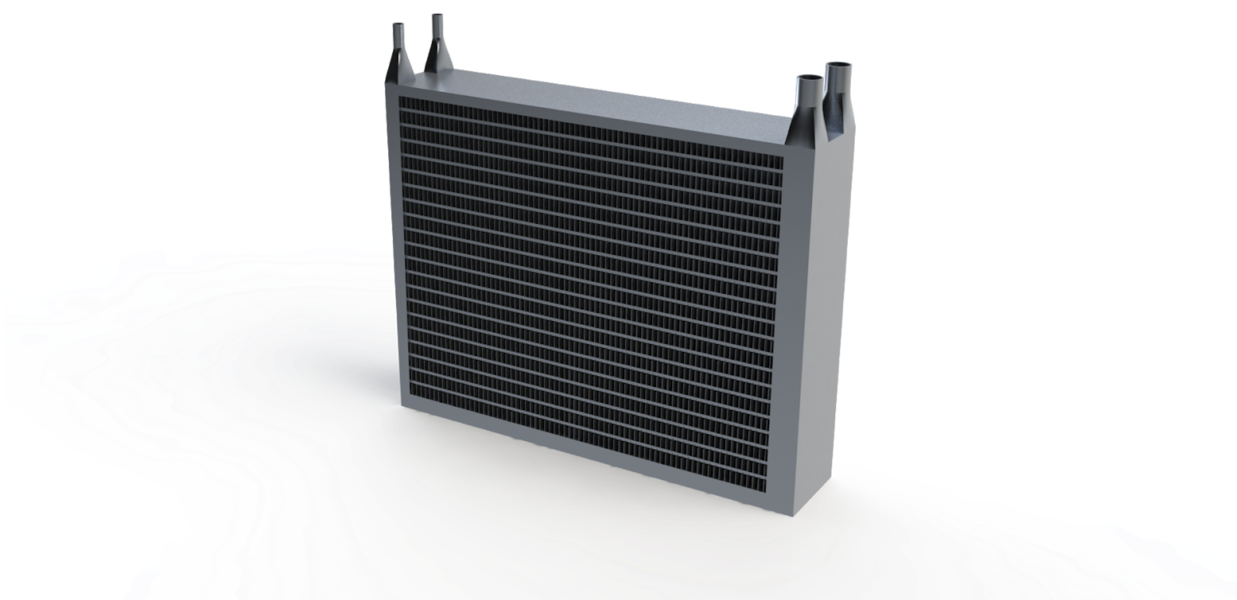


Figure 8.1. Model of the final heat exchanger (not simulated)

The approximate overall **dry weight** of the heat exchanger including fins is about **15.9 lbs**.

In **Figure 8.2**, a wireframe drawing was made to show more detail of the heat exchanger as well as overall dimensions. The width and height is 20 in and 15 in, respectively, which meets the 300 in^2 maximum frontal area requirement. The depth of the radiator is 4 in, featuring 2 columns of 0.1 in X 1.0 in (internal cross section) tubes, 20 tubes per column. The 2 columns are separated 1.5 in, which was chosen by analyzing the fin temperature contours. In this design, the heat exchanger is essentially 2 separate radiators which are welded together. In terms of manufacturing, this makes the spacing between the two tube columns very flexible. Scale-up is also simple. For applications with increased oil flow, a 3rd or 4th column can be added to maintain the oil flow through each tube around 20 ft/min. The inlet manifold, on the left side of the radiator, features a taper to achieve an even flow distribution to the tubes. The exit manifold was designed to be roughly 2x the volume of the inlet manifold so tube-to-tube flow interactions would be less pronounced and would be less likely to influence the flow distribution in the inlet manifold. No significant design studies were conducted to assess the exit manifold geometry.

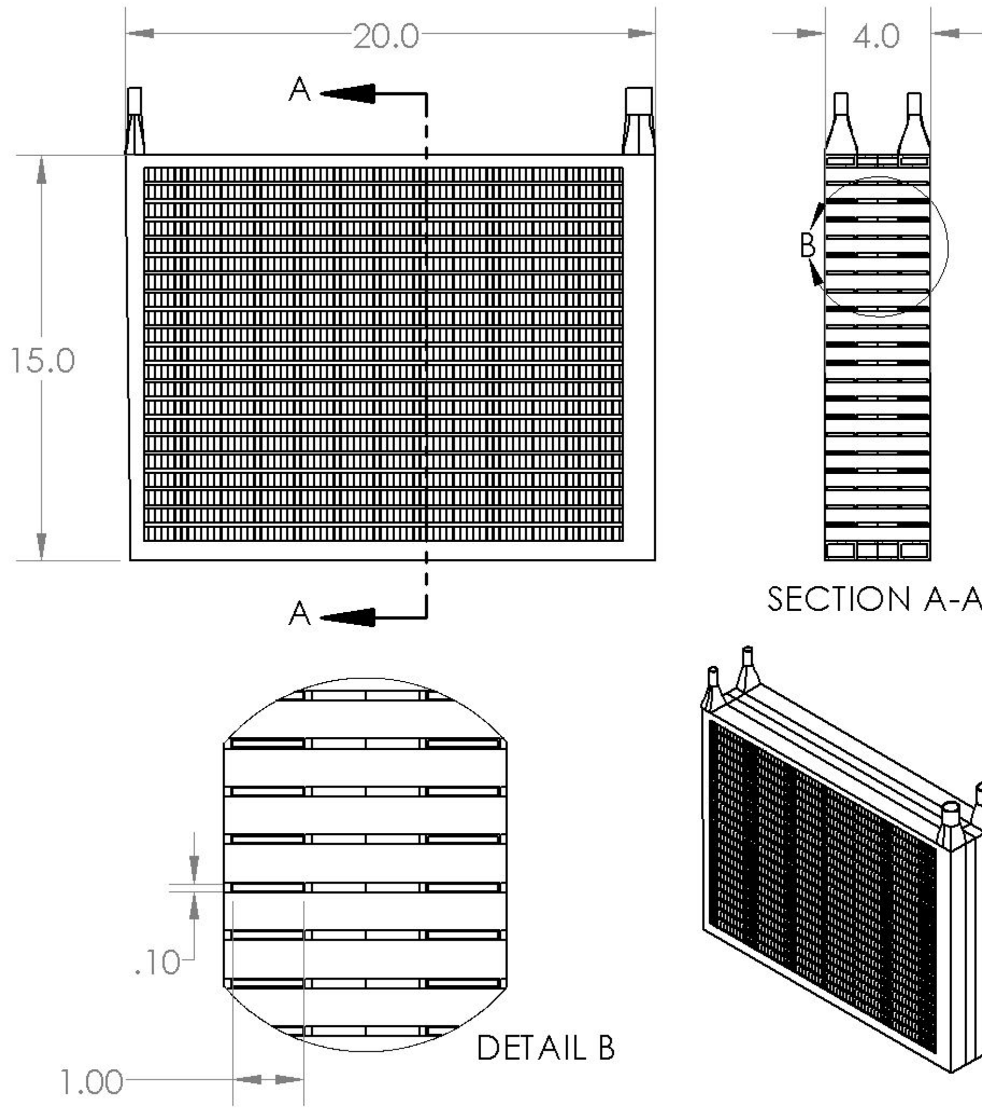


Figure 8.2. Model of the final heat exchanger (not simulated)

The cool air is brought to the heat exchanger via a straight duct with an immediate expansion.

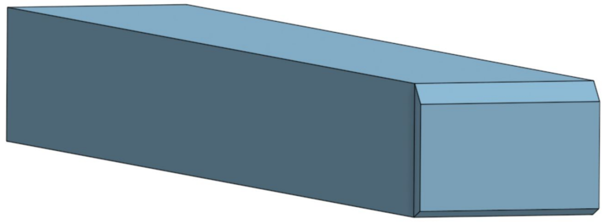


Figure 8.3. Geometry of the straight duct with an immediate expansion.

Alternatives include a straight duct without an immediate expansion and a curved duct with an immediate expansion (see *Design 1 and 3* in the *Air Duct Design* section). Compared to the alternatives, the straight duct seen in Figure 8.2 results in the highest outlet air velocity of 2058 ft/min from an initial minimum inlet velocity of 2200 ft/min. Maximizing outlet air velocity is critical for maximizing heat transfer.

The oil enters the heat exchanger with a volumetric flow rate of 4.0 gpm. A rectangular tube is used to carry the oil, which has dimensions of 1 in by 0.1 in (width X height). This volumetric flow rate is significantly reduced by splitting the oil flow into multiple tubes using a manifold design. The manifold design requires 40 tubes to decrease the oil velocity to 20 ft/min. The manifold is also tapered from an initial 0.5 in diameter to a 0.3 in end diameter.

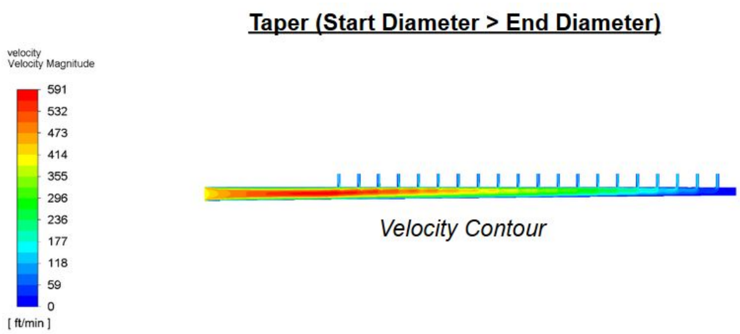


Figure 8.4. Velocity contour of the tapered manifold

The taper design allows the oil to be split into multiple tubes, each with similar velocities. The uniformity of the oil velocities allows the heat exchanger to behave more predictably, cooling the oil more uniformly.

The ideal fin thickness is determined to be around 0.015 in. For practical purposes, a 26 gauge aluminum sheet can be used, which has a thickness of 0.0159 in. The ideal fin spacing is 0.15 in, which can fit a total of 109 fins across a tube length of 18 in, but to minimize the weight of the heat exchanger, 0.25 in spacing can be used to fit 67 fins.

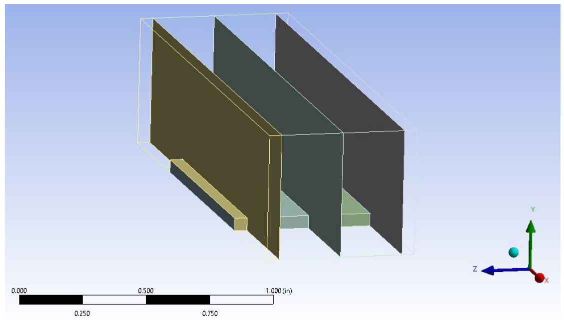


Figure 8.5. The fin and fin spacing around the oil carrying tube

The total number of fins can be increased up to 109 fins if additional cooling is required. With a 0.25 in spacing, the expected total number of fins that is required to drop the oil to 195°F is 33 fins. a 0.25 in spacing is more than enough to fit the required number of fins.

To verify pressure drop results in the three-fin model, hand calculations were used; specifically following the Darcy-Weisbach method. The pressure drop equation is the following:

$$\Delta P = \frac{\rho v^2}{2} f \frac{L}{D}$$

Where ρ is density, v is velocity, f is the Darcy friction factor, L/D is the length over diameter parameter, and ΔP is the pressure drop. For laminar flow (oil), the Darcy friction factor is:

$$f = \frac{64}{Re},$$

where Re is the Reynold's number of the flow. For turbulent flow (air), the friction factor can be calculated in various ways. The Haaland equation is one particular way that is easy to solve:

$$\frac{1}{\sqrt{f}} = -3.6 \log\left[\left(\frac{\varepsilon/D}{3.7}\right)^{1.11} + \frac{6.9}{\mathrm{Re}}\right],$$

where ε is the roughness of the flow surfaces. The pressure drop values from **Table 7.7** can be adjusted to the correct lengths in the final design to calculate the overall predicted pressure drop. The frictional loss calculated pressure drop values are compared to the CFD pressure drop values in **Table 8.1**. There is sufficient agreement between the hand calculations and the CFD results.

Fluid	Pressure Drop (psi) - Calc	Pressure Drop (psi) - CFD
Air	0.0065	0.014
Oil	0.34	0.11

Table 8.1. Total pressure drops - Hand calculations vs. CFD (excludes oil manifolds)

Design Verification via Hand Calculations

Log mean temperature difference (LMTD) method is used to extrapolate numbers of fins needed for the heat exchanger to reach design criteria, based on results from one fin model simulation. LMTD method dictates that, assuming constant air temperature, number of fins needed to reach a overall temperature drop is related to temperature drop over single fin in the following relationship:

$$N = \frac{\dot{Q}_{overall} \times \Delta T_{LM,3fins}}{\dot{Q}_{singlefin} \times \Delta T_{LM,overall}}$$

Where LM represents log mean temperature, and are calculated using:

$$\Delta T_{LM,3fins} = \frac{T_{oil,in} - T_{oil,out}}{ln(\frac{T_{air} - Toil,in}{T_{air} - T_{oil,out}})} \quad \Delta T_{LM,overall} = \frac{T_{oil,in} - T_{oil,final}}{ln(\frac{T_{air} - Toil,in}{T_{air} - T_{oil,final}})}$$

And heat flux is calculated using:

$$\dot{Q} = \dot{m} \times c_p \times \Delta T$$

The three fin model simulation yields a more significant and more reasonable drop in oil temperature. So this following calculation to find the number of fins is carried out using results from the 3 fin model simulation. In other words, heat flux and LMTD are calculated for across 3 fins, and used to find the number of fins needed.

$T_{oil,in}$ [F]	350
$T_{oil,out}$ [F]	**338**
T_{air} [F]	**108**
T_{air} out [F]	**132**
T_{air} avg [F]	**120**
$\dot{m}_{oil}$ [lbm/min]	1.86
$\dot{m}_{air}$ [lbm/min]	15.5
$c_{p,oil}$ [Btu/lbm F]	0.5
$c_{p,air}$ [Btu/lbm F]	0.24

Table 8.2. The variables to be used in the LMTD calculation - first iteration

Using the following parameters, for the first iteration in **Table 8.1**, the number of 0.25 in spaced 3 fin units required to reach the desired 195 °F oil temperature drop is 20, or 60 fins in total.

$T_{oil,in}$ [F]	350
$T_{oil,out}$ [F]	**339**
T_{air} [F]	**132**
T_{air} out [F]	**154**
T_{air} avg [F]	**143**
$\dot{m}_{oil}$ [lbm/min]	1.86
$\dot{m}_{air}$ [lbm/min]	15.5
$c_{p,oil}$ [Btu/lbm F]	0.5
$c_{p,air}$ [Btu/lbm F]	0.24

Table 8.3. The variables to be used in the LMTD calculation - second iteration

The exit average air temperature from the first iteration was used as the inlet air temperature for the second iteration since there are 2 columns of tubes in the heat exchanger design.

Since the width of the exchanger is 20 in, and leaving 1 in to each side for the inlet/outlet manifold and frames, there are 18in of path length for 0.25 in spaced fins, or $\frac{18in}{0.25\ in} = 72\,fins$.

Using the LMTD approach and fixing the number of fins to 72 fins, the exit oil temperature was solved for and the results are presented in **Table 8.4** below.

Exit Temp (deg F) - 1st Column	Exit Temp (deg F) - 2nd Column
184.7	192.0
Average oil temperature = **188.4**	

Table 8.4. Resulting oil exit temperatures using LMTD method

After combining the oil from the 1st column with the 2nd column via the exit manifold, the expected exit temperature should be the average of 184.7 deg F and 192.0 deg F, which is ~188.4 deg F.

To solve for efficiency of a cross-flow heat exchanger, the equations are:

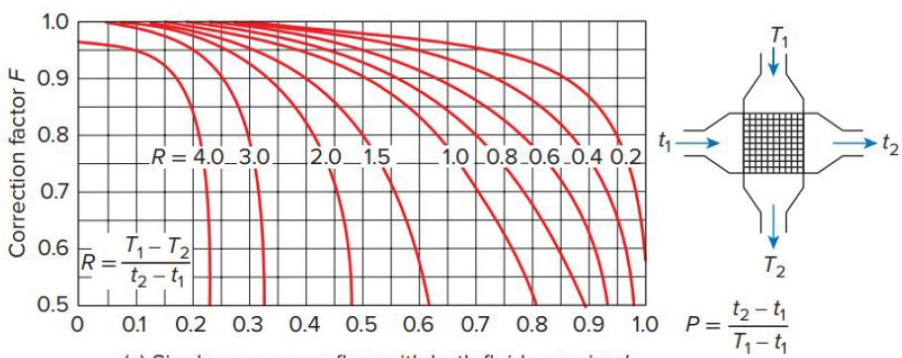
$$\eta = \frac{1 - e^{-\alpha}}{1 - \frac{m'_a Cp_a}{m'_b Cp_b} e^{-\alpha}} \text{ , and}$$

$$\alpha = U_i \pi D L \left(\frac{1}{m'_a Cp_a} - \frac{1}{m'_b Cp_b} \right)$$

where m’a and C p_a are the mass flow and heat capacity of fluid a. Fluid b properties are denoted the same way. Since heat is transferred from oil to the surrounding air, the oil is fluid a and the air is fluid b. U_i is the overall heat transfer coefficient of the model which needs to be calculated from the following equation:

$$m'c_p \Delta T = U_i A_i \Delta T_{LM} F$$

where A_i is the exposed surface area and F is a correction factor given by the graph below, specific to cross-flow heat exchangers. In this case, F is very close to 1, so 1 is assumed.



(c) Single-pass cross-flow with both fluids *unmixed*

The calculated efficiency is ~ 34%.

Summary of Results

Heat Exchanger Dimensions and Design

Overall dimension [width x length x depth]	15in x 20in x 4in
Material	Aluminum 6061
Total number of parallel tubes (oil)	40 tubes
Number of passes	2
Number of tubes per pass	20 tubes
Cross section area of parallel tubes (oil) [width x height]	1in x 0.1in
Distance between tubes (center to center)	0.675
Total Number of fins	1512
Number of fins per layer	72
Number of fin layers	21
Total fin area	6471 in^2

Heat Exchanger Performance

Mass	15.9 lbs
Efficiency	~34%

Worst Case Operation Rating:

Oil flow rate	4.0 Gpm
Oil inlet temperature	350 °F
Oil exit temperature	**188** °F
Oil manifold pressure drop	1.47 psi
Oil cores pressure drop	0.11 psi
Oil total pressure drop	**1.58 psi**
Air flow rate	2000 ft/min
Air inlet temperature	108 °F
Air exit temperature	**154 °F**
Air pressure drop from duct	0.012 psi
Air pressure drop from heat exchanger	0.014 psi
Air pressure drop from duct	**0.026 psi**

Design Benefits

The final heat exchanger design is lightweight and corrosion resistant through the use of aluminum instead of copper. The total dry weight of the heat exchanger including fins and the manifolds is ~ 16 lbs, which would be substantially lighter than a copper configuration. The design also has flexibility since it is essentially two heat exchangers stacked together. The spacing between tube columns can easily be adjusted. Scale-up or scale-down is also possible by stacking/removing units, allowing this design to be used for a wide range of applications. The corrosion resistance offered by aluminum compared to copper is not trivial. By using aluminum, the usable lifetime of this heat exchanger will be substantially more than that of copper.

Design Time Estimate

150-160 hours total