

### Design and Thermal Analysis of Hydraulic Oil Cooler by Using Computational Fluid Dynamics (CFD)

Ankush S. Patil\(^1\), H. S. Farkade\(^2\)

\(^1\) P.G. Student, Dept. of Mechanical Engineering, Govt. College of Engineering, Amravati, Maharashtra, India.
\(^2\) Assistant Professor, Dept. of Mechanical Engineering, Govt. College of Engineering, Amravati, Maharashtra, India.

**Abstract** - The hydraulic oil cooler come equipped with the cross flow heat exchanger cores that have a shroud with fans and other brackets and braces to secure the components into the reservoir. In application, heat to be rejected from the oil is flowing into the “core” is placed in the overheating reservoir which insures that the returning oil. The working fluid in the cross flow heat exchanger rejects its heat to the surrounding forced air respond reduction in the temperatures of oil. This may avoid the fall in viscosity of oil and the problem of overheating of oil. This is passive and efficient heat removal. As long as there is a temperature difference between the oil and the ambient air, the cross flow heat exchanger will remove heat from the oil, thus cooling it down. It is the process that provides the mechanism by which heat is “removed” from one medium to another. In the case of hydraulic oil coolers – heat is removed from oil into the ambient air.

**Key Words:** CFD, Parallel flow, Counter flow, LMTD, Heat transfer, Effectiveness.

### 1. INTRODUCTION

Heat exchangers are those devices which are used to transfer heat from hot fluid to cold fluid which are of either same or different phases. Heat exchangers are used in wide range for different types of industrial and domestic applications. Some of the heat exchangers are mixing type and some are non-mixing type. The difference between the mixing and non-mixing is that in non-mixing type the fluids are separated by metal wall. In non-mixing type as there is used a metal wall to separate the two different fluids, the heat transfer takes place by convection in each fluid and by conduction through the walls, so that in the analysis of heat exchanger, it is necessary to calculate overall heat transfer coefficient \( U \). Heat load depends on heat rejection required to keep engine surface at optimum temperature. Generally LMTD or \( \epsilon \)-NTU method is used to do heat transfer calculations of radiator. Both methods have its own advantages and preferred according to data availability. When radiator inlet and outlet temperature are known LMTD gives faster solution. When any of the temperature is unknown LMTD method undergoes iterations to find solution. In this case \( \epsilon \)-NTU is better. In this paper \( \epsilon \)-NTU method is described to do heat transfer calculations.

### 2. EXPERIMENTAL SETUP

We proposed to carry out on experimental set-up as shown in the figure 3.5. The experiment setup will be a close-loop experimental facility as shown in figure 3.5. The loop consisted of a hydraulic press bell jinning machine system, accumulator, and heat exchanger along with cooling fan. After treatment of SAE60 oil in hydraulic press bell jinning machine it will suddenly get heated up to 60°C. Then oil is recirculating throughout the close loop cycle. The

---

**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>( Re )</td>
<td>Reynolds number</td>
<td></td>
</tr>
<tr>
<td>( Nu )</td>
<td>Nusselt number</td>
<td></td>
</tr>
<tr>
<td>( Q )</td>
<td>Heat transfer rate</td>
<td></td>
</tr>
<tr>
<td>( \eta )</td>
<td>Thermal performance factor</td>
<td></td>
</tr>
<tr>
<td>( m )</td>
<td>Mass flow rate of fluid (Kg/s)</td>
<td></td>
</tr>
<tr>
<td>( D )</td>
<td>Hydraulic diameter (m)</td>
<td></td>
</tr>
<tr>
<td>( W )</td>
<td>Width of Helical Tape inserts (m)</td>
<td></td>
</tr>
<tr>
<td>( A )</td>
<td>Surface area of test section (m²)</td>
<td></td>
</tr>
<tr>
<td>( \eta_{fin} )</td>
<td>Fin efficiency</td>
<td></td>
</tr>
<tr>
<td>( \eta_0 )</td>
<td>Overall surface efficiency</td>
<td></td>
</tr>
<tr>
<td>( q )</td>
<td>Heat flux ((W/m²))</td>
<td></td>
</tr>
<tr>
<td>( N_f )</td>
<td>Number of fins per tube</td>
<td></td>
</tr>
<tr>
<td>( \mu )</td>
<td>Viscosity of fluid, Ns/m²</td>
<td></td>
</tr>
<tr>
<td>( \rho )</td>
<td>Density of fluid, kg/m³</td>
<td></td>
</tr>
</tbody>
</table>

**Greek symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \mu )</td>
<td>Viscosity of fluid, Ns/m²</td>
</tr>
<tr>
<td>( \rho )</td>
<td>Density of fluid, kg/m³</td>
</tr>
<tr>
<td>( \kappa )</td>
<td>Thermal conductivity of air (W/mK)</td>
</tr>
<tr>
<td>( \epsilon )</td>
<td>Heat transfer coefficient (W/m²K)</td>
</tr>
<tr>
<td>( To )</td>
<td>Outlet oil temperature</td>
</tr>
<tr>
<td>( T_1 )</td>
<td>Inlet oil temperature</td>
</tr>
<tr>
<td>( Ta_1 )</td>
<td>Inlet air temperature</td>
</tr>
<tr>
<td>( Ta_2 )</td>
<td>Outlet air temperature</td>
</tr>
<tr>
<td>( h )</td>
<td>Convective heat transfer coefficient (W/m²K)</td>
</tr>
</tbody>
</table>
recirculating hot oil at 60°C is stored at accumulator then it is flow toward the inlet of heat exchanger (radiator). After treatment of hot oil by the application of cooling fan whose temperature is at 40°C the hot oil temperature reduces up to 45°C. The heat from the hot oil is gained by the cold fluid air thus the temperature of oil gets reduced. After treatment the oil at 45°C is collected at the oil tank having capacity of 1400 litre. The oil at ambient temperature from the oil tank is further ready for the operation in hydraulic press bell jinning machine.

As For Counter Flow heat Exchanger,
The value of LMTD for the counter flow heat exchanger is as follows,

\[ \theta_1 = (65-55) = 10 \]
\[ \theta_2 = (45-40) = 5 \]

\[ \text{LMTD}(6m) = \frac{91-62}{\ln(10/5)} \ln(10/5) = 7.21°C \]

Now, For Cross flow heat exchanger directly takes the value of correction factor for single cross flow heat exchanger with both fluid are unmixed as below,

\[ \text{Correction Factor, } F = 0.85 \]

Overall heat Transfer coefficient,

\[ \frac{1}{U} = \frac{1}{h_i} + \frac{1}{h_c} \]

Internal Flow of Oil

Velocity,

\[ V_{oil} = \frac{m_{oil}}{c_{oil} \cdot A_{tube}} = \frac{0.0083}{0.864 \cdot 0.012} = 1.22 \text{ m/s} \]

Reynolds No.

\[ R_{oil} = \frac{c_{oil} \cdot D_{oil}}{\mu_{oil}} = \frac{864 \cdot 1.22 \cdot 0.01}{0.0071} = 723 \]

Nusselt No.

\[ N_{oil} = 0.023 \cdot R_{oil}^{0.8} \cdot P_{oil}^{0.4} = 0.023 \cdot 723^{0.8} \cdot 950^{0.4} = 26.10 \]

Heat transfer coefficient,

\[ h_{oil} = \frac{N_{oil} \cdot c_{oil} \cdot D_{oil}}{L_{tube}} = 96.4 \cdot 0.147 = 743 \text{ W/m²k} \]

External flow of Air

Velocity,

\[ V_{air} = \frac{m_{air}}{c_{air} \cdot A_{tube}} = \frac{0.225}{1.16 \cdot 0.0648} = 3 \text{ m/s} \]

Reynolds No.

\[ R_{air} = \frac{c_{air} \cdot D_{air}}{\mu_{air}} = \frac{1.16 \cdot 0.0648}{1.16 \cdot 0.01} = 87731 \]

Nusselt No.

\[ N_{air} = 0.664 \cdot R_{air}^{0.1} \cdot P_{air}^{0.5} = 0.664 \cdot 87731^{0.5} = 0.74 \cdot 3 \approx 178 \]

Heat transfer coefficient,

\[ h_{air} = \frac{N_{air} \cdot c_{air} \cdot D_{air}}{L_{tube}} = 28.54 \cdot 0.025 = 71.35 \text{ W/m²k} \]

Now, From above Equation the overall heat transfer coefficient is \[ U = 64.80 \text{ W/m²k} \]

To calculate the total surface area

We have,

\[ q = U \cdot A_s \cdot \text{LMTD} \cdot F \]

\[ A_s = \frac{3410}{64.80 + 7.21 + 0.58} = 8.678 \text{ m²} \]

Also,

\[ A_s = \pi \cdot d \cdot L \]
3.2 System Configuration

![Image](image1.png)

No. of tube, \( N = \frac{\text{Total length of tube}}{\text{Length of single tube}} \)

\( N = \frac{L}{l} = \frac{33.6 \text{ m}}{0.6} = 46 \)

3.3 Mathematical Model

The goal of the project is to use the analytical heat transfer process to determine outlet coolant and air temperatures and to compare this data with simulation results. The equations and calculations following are simply used to solve for both outlet temperatures. The initially assumed outlet temperatures and calculated outlet temperatures are then iterated to solve for the actual values for the outlet temperatures for air and oil.

\[ q = m_{\text{air}}c_{\text{p,air}}(T_{\text{air, out}} - T_{\text{air, in}}) = m_{\text{oil}}c_{\text{p,oil}}(T_{\text{oil, in}} - T_{\text{oil, out}}) \]

**Internal Flow of Oil**

The hot water from the engine travels through the tubes of the radiator.

**Area of Tubes,**

\[ \frac{4A_{\text{tube}}}{D_{\text{hydraulic}}} \]

The hydraulic diameter must be used because it is a non-circular cross section.

**Mean Temperature of Oil**

The average temperature of water must be calculated to find the fluid's material properties. The properties will be interpolated at this temperature. The properties that are needed are density, Prandtl number, thermal conductivity, dynamic viscosity, and specific heat.

**Velocity**

\[ \nu_{\text{oil}} = \frac{Q_{\text{oil}}}{N_{\text{tube}}A_{\text{tube}}} \]

The velocity of the water through each tube must be found to calculate the Reynolds number. The number of tubes is given by the chosen radiator.

**Reynolds Number**

\[ Re_{\text{oil}} = \frac{\rho_{\text{oil}}\nu_{\text{oil}}D_{\text{oil}}}{\mu_{\text{oil}}} \]

**Nusselt Number**

The Nusselt number was found as a constant for a rectangular cross section from Table 8.1 for fully developed laminar flow. The ratio of width over height of the tube is used in this table to determine the Nusselt number.

**Convective Heat Transfer Coefficient for Oil Flow**

\[ h_{\text{oil}} = \frac{N_{\text{oil}}K_{\text{oil}}}{D_{\text{hydraulic}}} \]

**External Flow of Air**

The air flows from the fan across the radiator tubes and through the fins utilizing convective heat transfer. In reality, the flow of air over the tubes will be slightly different due to the fluid flowing around the first tube before reaching the second tube, so calculating the heat transfer coefficient would be very difficult. To simplify the calculations, the flow is assumed to be the same over both tubes. Also, because the...
height to width ratio of the tubes is so small, the air will be assumed to be flowing on both sides of a flat plate.

Mean Temperature of Air
The average temperature of air must be calculated to find the correct material properties for later use. These properties are specific heat, thermal conductivity, kinematic viscosity, and Prandtl number.

Velocity
\[ v_{air} = \frac{Q_{air}}{A_{radiator} - (N_{tube}H_{tube}L_{radiator})} \]

Reynolds Number
\[ Re_{air} = \frac{\rho_{air}v_{air}W_{fin}}{\mu_{air}} \]

Nusselt Number
Looking at the geometry of the tubes, it can be assumed that the flow of air is similar to parallel flow over a flat plate. Since the flow never reaches the critical Reynolds number for a flat plate, \( Re = 5 \times 10^5 \), it is said to be laminar for the entire process.

\[ Nu_{air} = 0.664Re^{\frac{1}{2}}Pr^{\frac{1}{3}} \]

Convective Heat Transfer Coefficient for Air Flow
\[ h_{air} = \frac{Nu_{air}K_{air}}{W_{tubes}} \]

Fin Dimensions and Efficiency
The geometry of the fins on the radiator is sinusoidal. The troughs of the fins touch the lower adjacent tube and the peaks of the fins touch the upper adjacent tube. The fin efficiency equation takes into account the geometry of the fin and its dimensions to find the efficiency the fin will have.

\[ n_{fin} = \frac{\tanh(mL_c)}{mL_c} \]

Overall Surface Efficiency
The overall surface efficiency is needed for the external flow of air because the imperfections of the flow around the fins must be considered.

\[ n_c = 1 - \frac{N_{fin}A_f}{A_{fin}} (1 - n_{fin}) \]

Effectiveness-NTU Method
The Effectiveness-NTU method is used to find the effectiveness of the system. The overall heat transfer coefficient is needed. The surface efficiency is needed for the external flow of air because the imperfections of the flow around the fins must be considered. Using the convective heat transfer coefficients of both the internal and external flows, the UA is calculated. This value is then used to calculate the NTU.

Overall Heat Transfer Coefficient
\[ UA = \frac{1}{\frac{1}{h_{air}^{h_{air}}{external}} + \frac{1}{h_{air}^{h_{air}}{internal}}} \]

Number of Transfer Units
\[ NTU = \frac{UA}{\epsilon_{min}} \]

Effectiveness
The radiator utilizes a cross-flow single pass design where both fluids remain unmixed. This correlates to a specific equation to calculate effectiveness. However, this equation requires the heat capacity ratio, \( C_r \), to be equal to 1. The calculated heat capacity ratio is 0.455; therefore, the effectiveness is only a close approximation instead of the true value.

\[ \epsilon = 1 - \exp\left(\frac{[\frac{1}{2}NTU^{0.22})(\exp(-C_rNTU^{0.78})-1)]}{C_r} \right) \]

Heat Transfer Rate
The maximum heat transfer rate must be found to find the predicted heat transfer rate. Once this is known, the final output temperature of both the hot and cold fluid is calculated using a modified version of the initial thermal energy equation. These outlet temperatures must be iterated with the initially guessed outlet temperatures until the numbers are equivalent. The iterated outlet temperatures for both air and water are the theoretical values, which are used to compare with the experimental results.

Max Heat Transfer Rate
\[ q_{max} = C_{min}(T_{out,air} - T_{air,ini}) \]

Predicted Heat Transfer
\[ q_{predicted} = \epsilon q_{max} \]

Temperature Out
\[ T_{out,air} = 1 - \frac{q_{predicted}}{C_{air}} \]
\[ T_{air,ini} = 1 - \frac{q_{predicted}}{C_{air}} \]

3.4 Theoretical Calculation

<table>
<thead>
<tr>
<th>Table -1:</th>
<th>Basic Radiator Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>( L_{rad} )</td>
<td>( H_{rad} )</td>
</tr>
<tr>
<td>inch</td>
<td>inch</td>
</tr>
<tr>
<td>26.125</td>
<td>18</td>
</tr>
</tbody>
</table>
Table -2: Properties of Oil at Average Temperature

<table>
<thead>
<tr>
<th>ρoil</th>
<th>cp,oil</th>
<th>Proil</th>
<th>koil</th>
<th>µoil</th>
</tr>
</thead>
<tbody>
<tr>
<td>kg/m³</td>
<td>J/kgK</td>
<td>unitless</td>
<td>W/mK</td>
<td>kg/sm</td>
</tr>
<tr>
<td>864</td>
<td>2047</td>
<td>1050</td>
<td>0.147</td>
<td>0.0717</td>
</tr>
</tbody>
</table>

Table -3: Properties of Air at Average Temperature

<table>
<thead>
<tr>
<th>ρair</th>
<th>cp,air</th>
<th>Pairs</th>
<th>kair</th>
<th>vair</th>
</tr>
</thead>
<tbody>
<tr>
<td>kg/m³</td>
<td>J/kgK</td>
<td>unitless</td>
<td>W/mK</td>
<td>m²/s</td>
</tr>
<tr>
<td>1.16</td>
<td>1007</td>
<td>0.74</td>
<td>2.50E-2</td>
<td>1.19E-5</td>
</tr>
</tbody>
</table>

Table -4: Final Results of Radiator

<table>
<thead>
<tr>
<th>hoil</th>
<th>hair</th>
<th>U</th>
<th>NTU</th>
<th>ε</th>
<th>qpredicted</th>
</tr>
</thead>
<tbody>
<tr>
<td>W/mK</td>
<td>W/mK</td>
<td>W/mK</td>
<td>unitless</td>
<td>%</td>
<td>W</td>
</tr>
<tr>
<td>748</td>
<td>72.2</td>
<td>70.6</td>
<td>0.59</td>
<td>45</td>
<td>3786</td>
</tr>
</tbody>
</table>

Table -5: Theoretical Temp of Fluids through Radiator

<table>
<thead>
<tr>
<th>Toil,in</th>
<th>Toil,out</th>
<th>Tair,in</th>
<th>Tair,out</th>
</tr>
</thead>
<tbody>
<tr>
<td>K</td>
<td>K</td>
<td>K</td>
<td>K</td>
</tr>
<tr>
<td>338</td>
<td>321.25</td>
<td>313</td>
<td>328.02</td>
</tr>
</tbody>
</table>

4. CFD ANALYSIS

Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modelling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the results, discussions and conclusions.

4.1 Pre –Processor

Geometry

Heat exchanger is built in the ANSYS workbench design module. It is a counter-flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinates system within ANSYS. The design modeller opens as a new window as the geometry is double clicked.

4.2 Solver

1. Problem Setup

The mesh is checked and quality is obtained. The analysis type is changed to Pressure Based type. The velocity formulation is changed to absolute and time to steady state.
3. Materials

The create/edit option is clicked to add water-liquid and copper and stainless steel and galvanized iron to the list of fluid and solid respectively from the fluent database.

5. Boundary Conditions

The main boundary condition include, a mass flow rate inlet boundary condition where used in the inlet nozzles. The cylindrical shaped geometries are the wall. At the outlet, the pressure outlet (atmospheric pressure) boundary condition was used. And all other portions are considered as the wall boundary with convective heat transfer surfaces. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition.

Inlet
> Select inlet and change type to mass flow inlet
> Enter the mass-flow rate
> Enter the value of inlet gauge pressure = 1.5e5 Pa
> Provide the inlet temperature = 338K
> Change the option under Direction Specification to “Normal to boundary”

Outlet
> Select the outlet and change type to “Pressure-outlet”
> Enter the value for outlet gauge pressure = 0 Pa
> Change the option under Direction Specification to “Normal to boundary”

Wall
> Select wall and change type to “Wall”
> Select the material for the wall: Aluminum
> Choose convection as the heat transfer mechanism employed in wall.
> Enter the value for heat transfer coefficient as 92 W/m²K.
> Enter wall thickness as 1mm Enter the free stream temperature on wall

4.3 Post-Processor
At first the analysis is done by keeping inlet temperature of coolant and free stream temperature constant and temperatures are obtained for different mass flow rates. Varied input mass flow rates are given in the table.

Temperature Counter Plot

1. Mass flow rate 0.083kg/s
   Fig -7: Temperature contour for mass flow rate 0.083kg/s

2. Mass flow rate 0.093kg/s
   Fig -8: Temperature contour for mass flow rate 0.093kg/s

3. Mass flow rate 0.103kg/s
   Fig -9: Temperature contour for mass flow rate 0.103kg/s

4. Mass flow rate 0.113kg/s
   Fig -10: Temperature contour for mass flow rate 0.113kg/s

5. Mass flow rate 0.123kg/s
   Fig -11: Temperature contour for mass flow rate 0.123kg/s
5. DATA VALIDATION

Comparison of analytical and simulation results

Table 7: Reading of temp for various Mass Flow Rates

<table>
<thead>
<tr>
<th>Mass flow rate (Kg/s)</th>
<th>Mass flow rate (Kg/s)</th>
<th>Oil inlet temp (K)</th>
<th>Oil outlet temp (K)</th>
<th>Air inlet temp (K)</th>
<th>Air outlet temp (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.083</td>
<td>0.225</td>
<td>338</td>
<td>322.62</td>
<td>313</td>
<td>326</td>
</tr>
<tr>
<td>0.093</td>
<td>0.225</td>
<td>338</td>
<td>324.84</td>
<td>313</td>
<td>325</td>
</tr>
<tr>
<td>0.103</td>
<td>0.225</td>
<td>338</td>
<td>328.24</td>
<td>313</td>
<td>321</td>
</tr>
<tr>
<td>0.113</td>
<td>0.225</td>
<td>338</td>
<td>329.38</td>
<td>313</td>
<td>320</td>
</tr>
<tr>
<td>0.123</td>
<td>0.225</td>
<td>338</td>
<td>331.92</td>
<td>313</td>
<td>318</td>
</tr>
</tbody>
</table>

5.1 Discussion of Results

The relationship of above graph shows that increase in mass flow rate of oil results in increase in heat transfer rate. The nature of graph is linear for flow rate range from 0.083 Kg/s to 0.103 Kg/s. After that point because of sudden increase in mass flow rate the nature of graph diverts.

Chart 1: Mass flow rate v/s. heat transfer rate

Comparison shows that both results closely matched with each other. Thus theoretical thermal analysis of radiator using ε-NTU method is validated using simulation approach.

Table 8: Analytical results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total heat transfer</td>
<td>Q</td>
<td>W</td>
</tr>
<tr>
<td>Oil outlet temperature</td>
<td>Toc</td>
<td>K</td>
</tr>
<tr>
<td>Air outlet temperature</td>
<td>Toa</td>
<td>K</td>
</tr>
</tbody>
</table>

Table 9: Simulation results

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total heat transfer</td>
<td>Q</td>
<td>W</td>
</tr>
<tr>
<td>Oil outlet temperature</td>
<td>Toc</td>
<td>K</td>
</tr>
<tr>
<td>Air outlet temperature</td>
<td>Toa</td>
<td>K</td>
</tr>
</tbody>
</table>

Chart 2: shows that as the mass flow rate increases the overall heat transfer rate (U) increases.

The heat transfer coefficient is maximum i.e. 110 W/m² °K at mass flow rate of 0.123 Kg/s. The nature of graph is linear up to flow rate of 0.093 Kg/s. It get diverted at this point and increases sudden up to flow rate of 0.103 Kg/s. From this point graph shows slightly reducing nature up to flow rate of 0.113 Kg/s and again graph shows linear nature.

6. CONCLUSIONS

A CATIAV5r18 and CFD package (ANSYS FLUENT 13.0) was used for modelling and CFD study of heat transfer characteristics of a hydraulic oil cooler by the application of heat exchanger for counter flow. The objectives of the project, to find and compare the inlet and outlet temperatures of the fluids in the radiator were accomplished successfully. Several important conclusions could be drawn from the present simulations and would be presented as follows.

- It is found that overall heat transfer coefficient increases with the increase in mass flow rate. For maximum mass flow rate (0.123 Kg/s) it is found to be increased up to 110 W/m² °K which more than theoretical heat load hence system is safe and will not overheat.
- The capacity ratio and effectiveness is increased with the mass flow rates. This shows that the existing cooler is more efficient at higher flow rates. The highest effectiveness is found as 0.725.
- Ansys Fluent is good CFD program to to simulate the heat transfer cases.
REFERENCES


