American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-12, Issue-10, pp-202-217
www.ajer.org
Research Paper Open Access

# Identify Fundamental Properties in Thermal System Design

Menwer F. Alenezi[1] and MohammadF.KH .Alenezi[2]

[1]*The Public Authority for Applied Education and Training , Vocational Training Institute, Automotive Mechanic Department ,Kuwait.*

[2]*The Public Authority for Applied Education and Training , Vocational Training Institute, Automotive Mechanic Department ,Kuwait.*

***Abstract***

*This study focuses on the basic principles of thermodynamics and uses it towards the application in thermal system design to identify fundamental properties such as Nusselt number and the heat transfer coefficient. It is divided into part 1 and part 2. For part 1, the methods used for calculations of the heated cylinder were analytical calculations by hand, a code developed for replicability in MATLAB R2019, and a thermal simulation was run on ANSYS to verify our results. Moreover, the same methods were applied to part 2, the heat exchanger. Part 1 yielded a heat transfer coefficient of 48.8977 W/m2*K, while the ANSYS calculated it to be 42.67462 W/m2*K with a 3.11 standard deviation and 2.2 confidence interval. Part 2 includes the heat convection of 6.3362W in the analytical process; on the other hand, the simulation a 5.5298W projecting a standard deviation of .4 and a confidence interval of .285. Amid the pandemic crisis, novel methods in computational and mathematical modeling were used.*

---

Date of Submission: 15-10-2023 Date of acceptance: 31-10-2023

---

## I. Introduction

In this report, a heated cylinder and heat exchanger will be analyzed and simulated in two separate parts: part one and part two. Before moving forward some concepts must be defined. Heat transfer, any or all of kinds, is considered a mechanism that conveys energy and entropy from one location to another. The specific mechanisms are usually referred to as convection, thermal radiation, and conduction . Heat or energy transfer from a solid surface to fluid results from the fluid motion along the surface; this can be considered as convection. Convection may also be forced where the motion of the fluid is caused by external means such as a fan or a pump pushing the fluid and pressure differences. The process of convection occurs when the heat is transferred from the solid surface to the fluid through diffusion, then by advection, which is when the bulk fluid motion carries the transferred heat away from the solid surface.

Convection is a necessary process that should be taken into consideration, especially for protecting appliances. A good example of convection is air-cooled engines found in cars. When engines run for a prolonged time, they may overheat which causes the water in the jackets to heat. When this occurs it is vital to cool the engine .For this to work the water must be cooled as well. When the water heats, it flows into the pipes and is cooled down by fans. When the water cools, it is pumped into the engine to cool it down following the forced convection principle. The air-cooled engine has a simple design and forced convection with guided aerodynamic convection.

Heat exchangers are devices used to facilitate the transfer of heat between two fluids without mixing them through convection and a small, sometimes negligible, conduction. Heat exchangers are used in applications where the cooling or heating of a fluid is required. There exists many configurations for this device that vary in flow direction to number of pipes, inlets, and outlets and even the inclusion of elbows and other fitting. For the purposes of this project the water to water flow in a counterflow heat exchanger is observed. The fundamental concept of the Counterflow HE is that hot fluid and cold fluid enter the device from opposite ends.

In this project forced convection is studied and analyzed externally and internally. To understand this concept, analytical and modeling methods using the software *Fluent Ansys and MATLAB* were used. Results acquired from the analytical portion were compared to those from *ANSYS* . It is important to note that the *ANSYS* and analytical work was replicated by at least three team members in each run to ensure preciseness.

**Theoretical background**

When thinking of the principles of heat transfer, force convection is a crucial function. Forced convection moves or “forces” the fluid to increase the heat transfer, the components that can force the fluid through a system can be a fan, pump, or any type of suction device. When heat transfer is mentioned, the idea of heat rising means that fluid that is hot is less dense than the same fluid as if it was cold, but that is not always the case. Hotter material naturally ends up on top of the cooler material due to higher buoyancy of the hotter material, determined by density.

Convection is a method of heat transfer that is expressed by newton’s law of heating and cooling. Convection is (qconv)=h(Ts-T∞), where (Ts) is the initial temperature, and (T∞) is the final temperature of the material through a constant (h), which is the heat transfer coefficient. When convection is being forced by pushing cooled or heated air from one place to another, it can change (h) to heat or cool an object a lot quicker.

Based on that theory, an example of forced convection can be as simple as a ceiling fan. On a ceiling fan, the blades are angled, which forces that air down so when cool air is desired, the fan rotates counterclockwise, which mixes warm air and forces cool air downwards, creating a downdraft. A ceiling fan can also be used to pull cool air up and force the warm air down by switching the fan to rotate clockwise, causing an updraft.

There are different types of heat exchangers that operate differently based on the direction of the flow of the fluid within the heat exchanger. Two types of heat exchangers that are the most common are counter flow and parallel flow, and for the project, a counter-flow heat exchanger was the desired exchanger. The counter-flow heat exchanger operates by having a stream of one of the fluids in the opposite direction of the flow of the other fluid compared to the parallel where the fluid flows in the same direction. Since counter flow heat exchangers have the cold fluid going through one side and the hot fluid through the other, it is more efficient because it minimizes the thermal stresses through the exchanger by the temperature difference being uniform between the two fluids, meanwhile the parallel flow exchanger have large temperature differences at the ends which causes large thermal stresses.

**Analytical Set up (Part 1)**

For the first part of the project, a heated cylinder with a surface temperature of 80°C experiences forced convection when air at 25°C flows over it at a velocity of 2.5 m/s. The forced convection is to be determined. In order to determine the convection, the heat transfer coefficient and area have to be calculated. Using the Nusselt number, the heat transfer coefficient can be acquired.

To begin the analytical calculations, it is important to find the properties of air at the film temperature. To find the film temperature, the temperature of the cylinder and the air is added and divided by two resulting in 52.5°C

Using the film temperature and interpolation, the properties of air were acquired being the following:

Density (ρ)= 1.08375 kg/m$^3$
Specific Heat (Cp) = 1007 J/kg*K
Thermal Conductivity (k) = 0.0275325 W/m*K
Thermal diffusivity (α) = 2.5233*10$^{-5}$ m$^2$/s
Dynamic Viscosity (μ) = 1.9743*10$^{-5}$ kg/m*s
Kinematic Viscosity (ν) = 1.8234*10$^{-5}$ m$^2$/s
Prandtl Number (Pr) = 0.72215

Using the properties of air, Reynolds number and Prandtl number can be calculated to later solve for the Nusselt number.

Calculating Reynolds Number:

$$Re = \frac{\rho VD}{\mu} = \frac{(1.08375\ kg/m^3)\ (2.5m/s)\ (0.01m)}{1.9743*10^{-5} kg/m*s} = 1372.32$$

Calculating Prandtl Number:

$$Pr = \frac{Cp\mu}{k} = \frac{1007\ J/kg*K\ 1.9743*10-5\ kg/m*s}{0.0275325\ W/m*K} = 0.7221$$

Using Table 7-1 from Heat and Mass Transfer Fundamentals and analyzing the range of Reynolds number, the equation used for Nusselt number is the following:

$$Nu = 0.683 Re^{0.466} Pr^{1/3}$$

Now, calculating the Nusselt number using Reynolds number and Prandtl number:

$$Nu = 0.683 Re^{0.466} Pr^{1/3} = 0.683(1372.32)^{0.466}(0.7221) = 17.7558$$

Nusselt number of a cylinder can be defined as:

$$Nu_{Cyl} = \frac{hD}{k}$$

Where h is the heat transfer coefficient.

Using the equation above for Nusselt number, the heat transfer coefficient is calculated to be:

$$h = \frac{Nu_{Cyl}k}{D} = \frac{(17.76)(0.0275325\ W/m*K)}{0.01m} = 48.8977\ W/m^2*K$$

Using the area of the cylinder:

$$As = 2\pi \frac{d}{2} h + 2\pi(\frac{d}{2})^2 = 2\pi(\frac{.01}{2})0.07 + 2\pi(\frac{.01}{2})^2 = 2.356*10^{-3}\ m^2$$

Having solved for the area and the heat transfer coefficient, now the convection can be determined with the following equation:

$$Q_{conv} = hAs(T_s - T_\infty) = (48.8977\ W/m^2*K)(2.356*10^{-3}\ m^2)(80°C - 25°C) = 6.3362W$$

**MATLAB (Part 1)**

A MATLAB code program was developed for the first part of the project, reproducibility and ease of access when changing dimensions and conditions.

Code:

```
%Project 1 Calculations Project 1
Ts=80 ; %Surface Temp DegreeCelcius
Tinf=25; %Temperature Infinity Degree Celsius
Tf=(Ts+Tinf)/2;
s1=['Film temperature is ',num2str(Tf),' *C '];
disp(s1)

%Properties of air at Tf
p=1.08375; %density (rho) kg/m^3
cp=1007; %specific heat J/kg*K
k=.0275325;%thermal conductivity W/mK
a=2.5233*10^-5;%thermal diffusivity
u=1.9743*10^-5;%dynamic viscosity m^2/s
v=1.8234*10^-5; %kinematic viscosity m^2/s
```

```
%Additional Properties
v1=2.5; % velocity m/s
d=.01;% diameter m
height=.07;%height meters

%Reynolds Number
Re=(p*v1*d)/(u);
s2=['The Reynolds Number is ',num2str(Re),' . '];
disp(s2)
%Nusselt Number
Nu=(.683*Re^.466)*(Pr^(1/3));
s3=['The Nusselt Number is ',num2str(Nu),' . '];
disp(s3)

%Heat Transfer Coefficient
%NuCyl=(h*d)/k
h=(Nu*k)/d;
s4=['The heat transfer coefficient ',num2str(h),' W/m^2*k. '];
disp(s4)

%Surface Area of Cylinder
As=(2*pi*(d/2)*height)+(2*pi*(d/2)^2);

%Q Convection
QConv=h*As*(Ts-Tinf);
s4=['The Q convection ',num2str(QConv),' W. '];
disp(s4)

%end
```

Output:

```
>> project1
Film temperature is 52.5 *C
The Reynolds Number is 1372.3218 .
The Prandtl Number is 0.7221 .
The Nusselt Number is 17.7558 .
The heat transfer coeficient 48.8862 W/m^2*k.
The Q convection 6.3352 W.
```

**Analytical Set up (Part Two)**

For part two (*Counterflow Heat Exchanger*) of the project a similar approach was used by doing analytical calculations by hand and then modeling and running a simulation using *Fluent Ansys* to determine the accuracy of our analytical calculations. These handwritten calculations can be found in figures 21 & 22 of the appendix.

The counterflow HE that is the object of study had two fluid streams. An inner hot flow of water with a mass flow rate of 0.034 kg/s and an inlet temperature of 70 C. This inner flow is through a copper pipe with an inner diameter of 8.3 mm and an outer diameter of 9.5 mm. The annular flow for this counterflow HE is water that has an inlet in the opposite direction of the hot fluid. The annular flow has an inlet temperature of 20.0 C, a mass flow rate of 0.017 kg/s, and an outer diameter of 14.0 mm. The pipe has a length spanning 1320 mm or 1.32 m.The following equations and some slight variations were used in the analysis.

$$\frac{1}{UA_s} = R_{total} = \frac{1}{h_i A_i} + \frac{\ln\left(\frac{D_o}{D_i}\right)}{2\pi k L} + \frac{1}{h_o A_o}$$

$$Re_D = \frac{4\dot{m}}{\pi D_i \mu}$$

$$Nu_D = 0.023 Re_D^{(0.8)} Pr^{0.4}$$

$$h = Nu_D \frac{k}{D_i}$$

$$C_h = \dot{m}_h c_{ph}$$

$$C_c = \dot{m}_c c_{pc}$$

$$C_r = C_{min}/C_{max}$$

$$\dot{Q}_{max} = C_{min}(T_{h,in} - T_{c,in})$$

$$NTU = \frac{UA_s}{C_{min}}$$

$$\varepsilon = \frac{1 - \exp\left[-NTU(1 - C_r)\right]}{1 - C_r \exp\left[-NTU(1 - C_r)\right]}$$

$$\dot{q}_{actual} = \dot{Q}_{max}\varepsilon$$

$$\dot{q}_{actual} = \dot{m}_c c_{pc}(T_{c,out} - T_{c,in})$$

$$\dot{q}_{actual} = \dot{m}_h c_{ph}(T_{h,in} - T_{h,out})$$

**As**- is the surface area of heat transfer
**Rtotal**-is the total thermal resistance
**h**- is the convection heat transfer coefficient
**Re** -is the reynolds number
**D**- is diameter either inner or outer
**ṁ**- is mass flow rate
**Nu**- is nusseltnumber
**U**- is the overall heat transfer coefficient
**C**- is the heat capacity rate
**Q**- is the heat transfer rate
**T**- is temperature
**NTU**- is the number of transfer units
**ε**- is the effectiveness of the HE

The properties of water were used from the property tables of saturday water @ 70C and 20C.

The analysis portion involves finding both outlet temperatures, the NTU, and effectiveness and the overall convection coefficient, U. The Ansys part involves modeling the problem conditions to visualize the temperate and pressure contours as well as an xy plot of both parameters. There exists a few methods for fully analyzing HE. The 2 methods that were used are the *log mean temperature difference LMTD* and the effectiveness-NTU Method. Most of the terminology used in heat transfer and fluid flow problems is relevant when considering a HE problem.

It is important to note that during the analytical process the heat conduction resulting from the copper wall was considered. The water in this case is assumed to be pure water and the copper tube is considered to be free of friction and impurities. This allows for the friction factor and fouling factor to not be considered. The entire handwritten calculations can be found in figures 24 & 25 in the appendix

**Model Setup (Part One)**

Modeled the physical system for the heated cylinder using ANSYS Fluent. First, the geometry was defined, mesh was added, and formatted the setup in order to plot,and solve for the results.

Geometry:

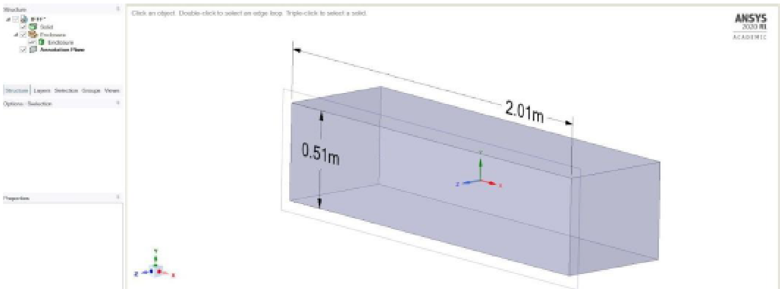


Figure 1. Fluid Domain for the Cylinder

The geometry consists of a cylinder with a diameter of 0.01 and a length of 0.07, the dimensions being in meters, enclosed in a rectangular block. The enclosure acts as a fluid domain for the simulation, and though it is not specified, its dimensions have to be reasonable and the enclosure was determined to be 0.5 * 0.5 * 2 in meters (width * height * length) . The cylinder is located at the center of the block. The length of the cylinder is directed in the z-axis.

Meshing:

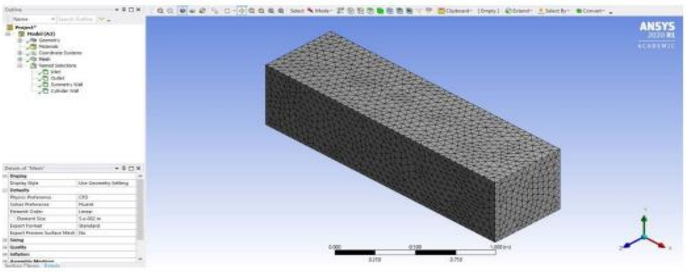

Figure 2. Meshed Enclosure

Meshed the enclosure using an element size of 0.05 meters. The final mesh has a total of 27250 nodes and 119942 elements.

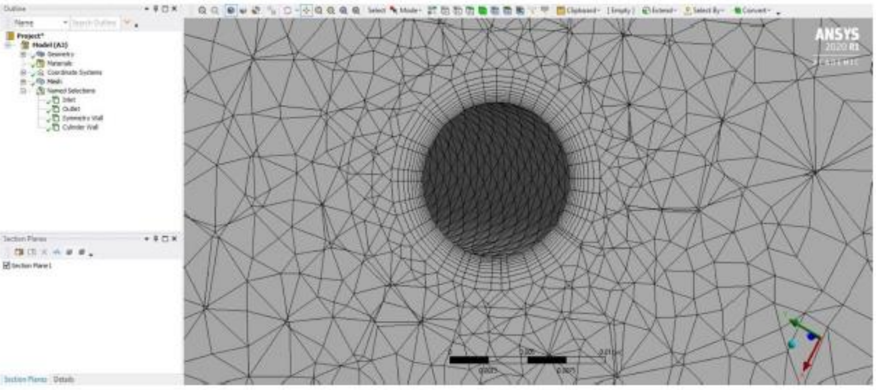


Figure3. Inflation

Then, inflation was created in the interest area of the domain by selecting the boundary wall of the cylinder.

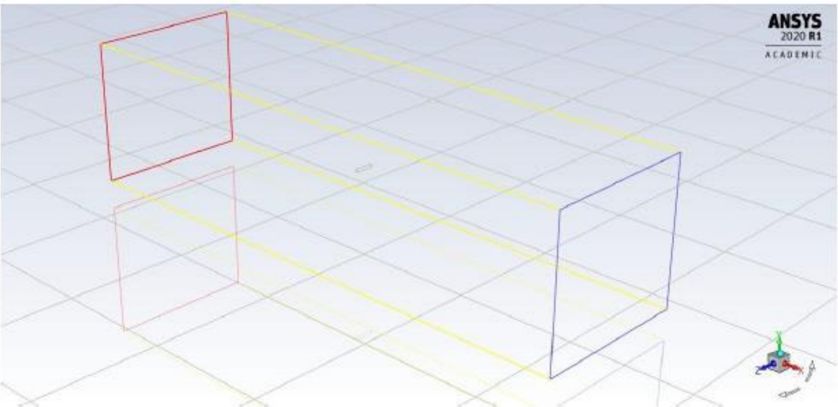


Figure 4. Assigning inlet and outlet

It was important to assign the faces of the enclosure as an inlet, outlet and wall. In Figure 4, the blue section represents the inlet, as red represents the outlet. The yellow represents the enclosure walls, and the small white region represents the cylinder wall.

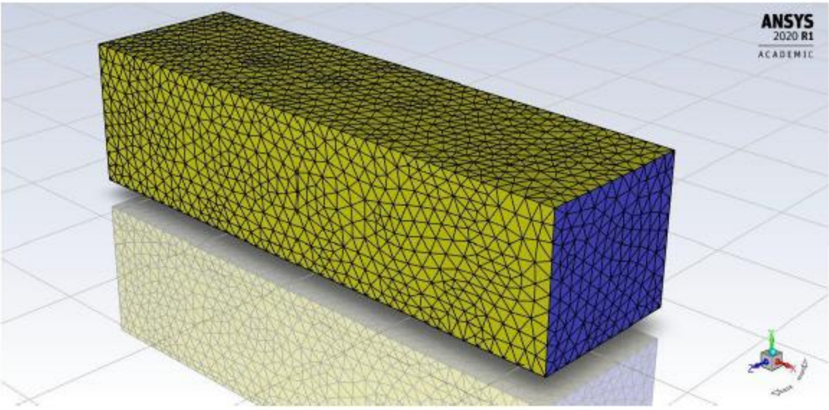


Figure 5. Defining Conditions

Materials, temperature, velocity, viscosity, density, and boundary conditions were defined and ran simulations of 200 iterations which can be shown in Figure 6 below.

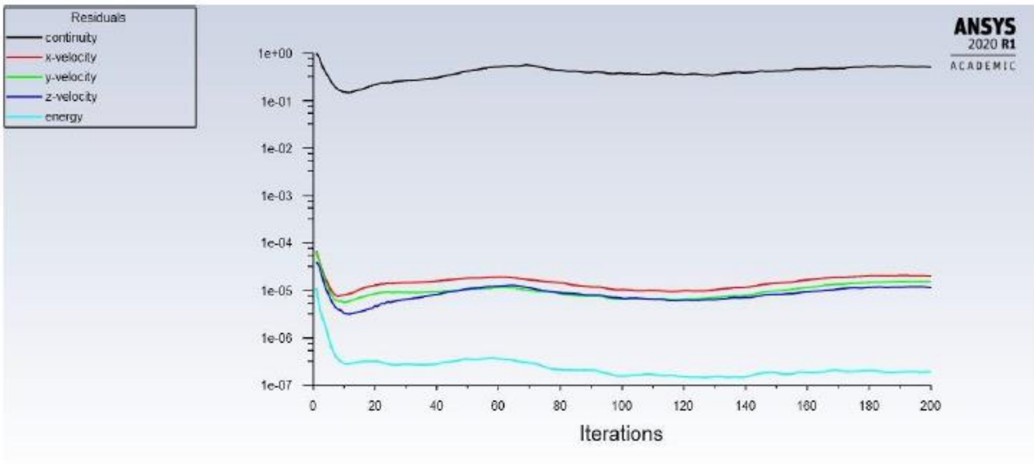


Figure 6. Iterations

Plots:

Lastly, after finishing the definitions and set up, plots of the contours were generated which can be found on the Results and Discussion portion.

**Model Set up (Part Two)**

For part two, a heat exchanger was created and simulated using ANSYS Fluent. The set up was similar to the cylinder. First the geometry was created and meshed. After setting the given conditions and parameters, the plots and results were solved for.

Geometry:

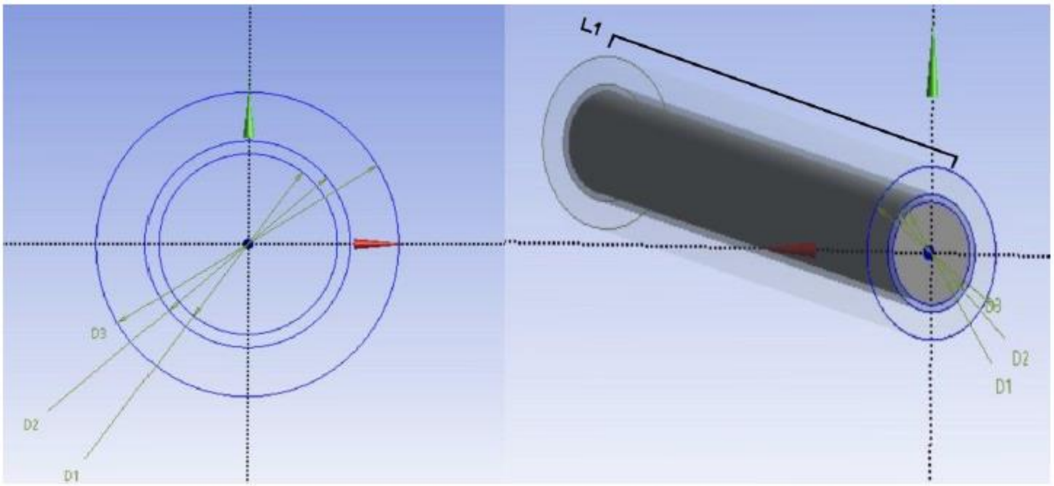


Figure 7. D1= 8.3mm D2=9.5mm D3=14mm L1=1320mm

This heat exchanger consisted of an inner tube with a diameter of 8.3 mm, outer pipe diameter of 9.5 mm, and the outer tube with a diameter of 14 mm. The inner tube was extruded with a length of 1320 mm. When extruding the outer pipe and outer tube, the extrusion operation had to be "add frozen," so three different bodies and parts are created.

Mesh:

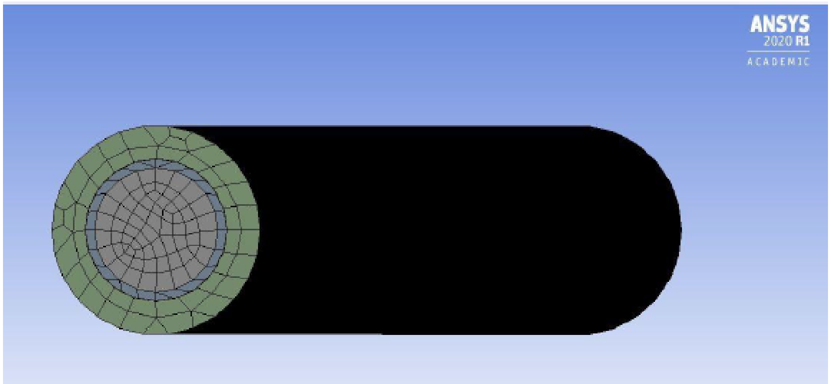


Figure 8. Meshed Heat Exchanger

After meshing the heat exchanger, the nodes and elements gave values of 177,464 and 128,720. In the mesh the outer and inner temperatures were assigned. The interfaces were also selected to show the two different fluid zones separated by a wall. The wall separating must be defined so the fluid can interact with the other fluid.

Setup:

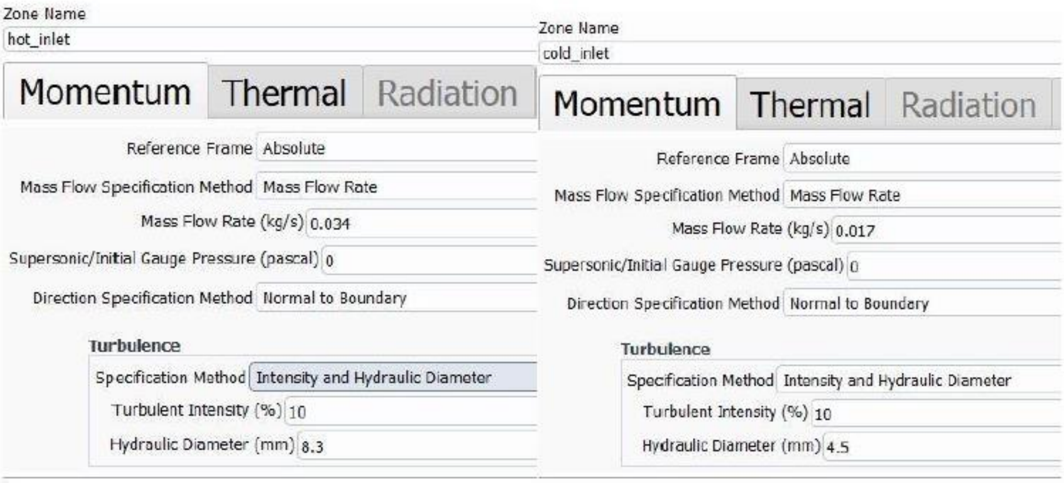


Figure 18. Hot and Cold Inlet Boundary Conditions

In the setup, different parameters were stated to set up our case. The first thing that needed to be done was creating two different mesh interfaces. The original materials were changed from air and aluminum to water and copper. In the boundary conditions, the cold and hot inlet mass flow rates are changed to .017 kg/s and .034 kg/s. This section is also where the cold and hot inlet temperatures are stated as 20C and 70C. These boundary conditions are the most important part of the set up. Once all parameters were stated, a simulation of 100 iterations was calculated to help show the results. For the results portion, to see the temperature contours, an iso-surface was created with the mesh and X-coordinate named center plane. To display the temperature, position plot, two lines were created, one in the cold and another in the hot region. (figure something) To show the problem as a counter flow, the boundary conditions had to change. A new Fluent was created to make these changes. Once those are changed, the same steps when solving for the parallel flow are followed. This gives the results of (figure showing plots etc).

**Results and Discussion**

For part one of the project, the analytical and simulated results were relatively close to each other. In the analytical process, the heat transfer coefficient was calculated to be 48.8977 $W/m^2*K$, while the ANSYS calculated it to be 42.67462 $W/m^2*K$ for the simulation.

The result for the Nusselt number from the simulation had to be analytically calculated using the heat transfer coefficient that the simulation measured. Using the the equation for Nusselt number for a cylinder, ($Nu_{Cyl} = \frac{hD}{k}$), and plugging in the new value for (h) the result was 15.50. In the analytical process, the Nusselt number had to be calculated first to solve for the heat transfer coefficient. The approach was different, the equation for the Nusselt number for a cylinder could not be used and so the Nusselt number had to be determined using an equation that depended on the range of the Reynolds number. Still, the Nusselt number for the analytical process was a relatively close number as it resulted being 17.7558.

In order to compare the thermal convections, the convection in the simulation had to be analytically calculated since the results on ANSYS only gave total heat transfer. Again, using the heat transfer coefficient that was calculated in the simulation, the heat convection can be acquired. Using the equation for heat convection ($Q_{conv} = hA_s(T_s-T_\infty)$) and plugin in the value for (h), the heat convection from the simulation was 5.5298W, while it was determined to be 6.3362W in the analytical process.

The following are the contour results for part one:

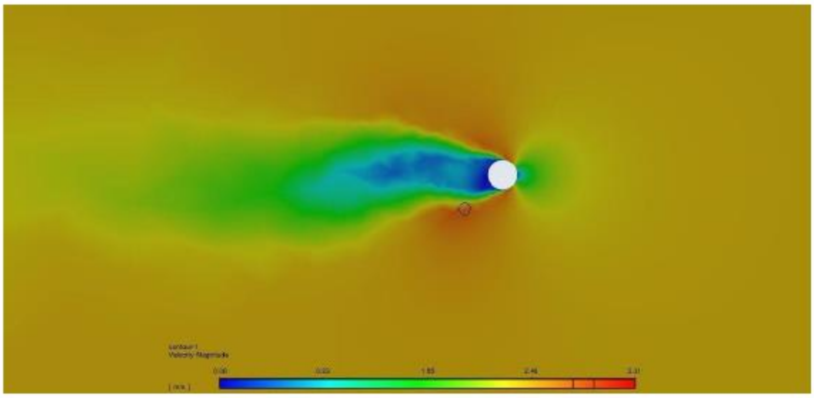

Figure 9. Contour of Velocity Magnitude (m/s)

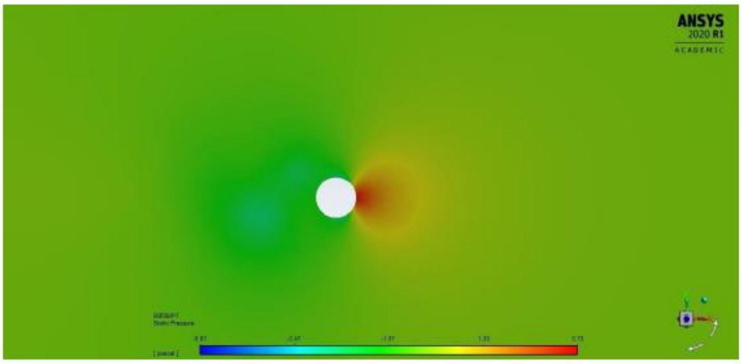


Figure 10.Contour of Static Pressure (pascals)

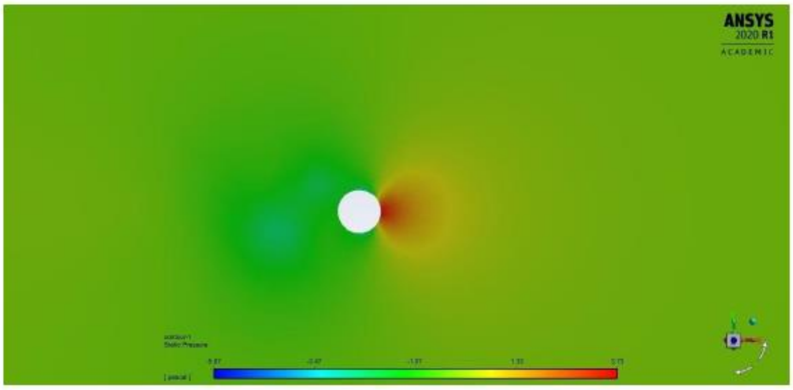


Figure 11. Contour of Static Pressure (pascals)

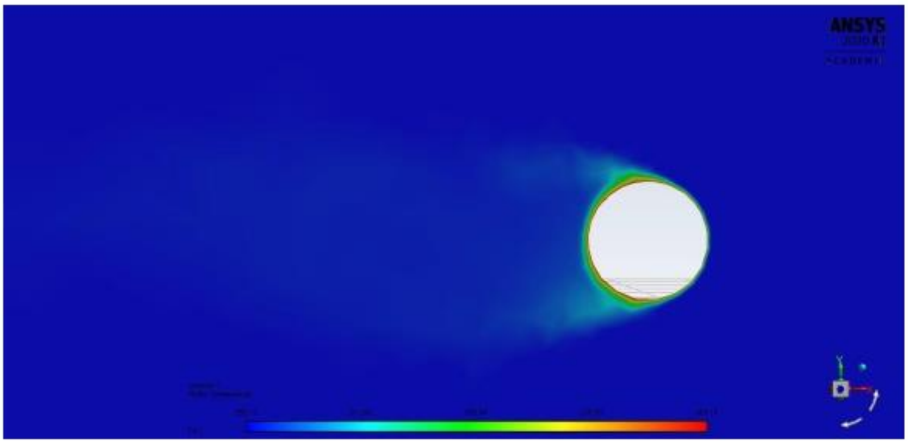


Figure 12. Contour of Static Temperature (k)

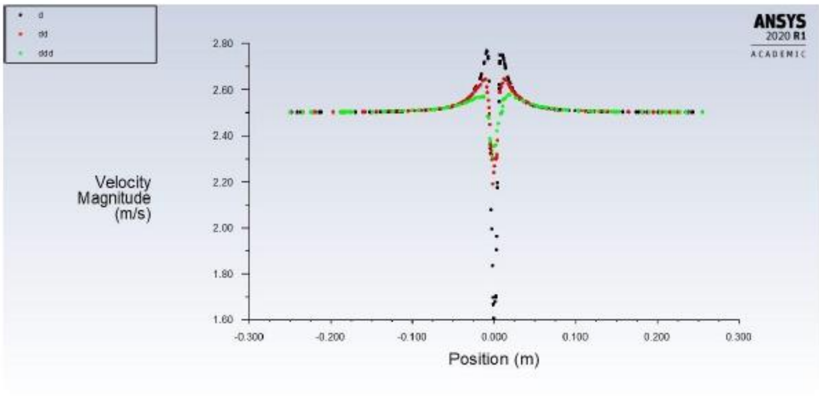


Figure 13. Velocity plot for D, 2D, and 3D downstream locations.

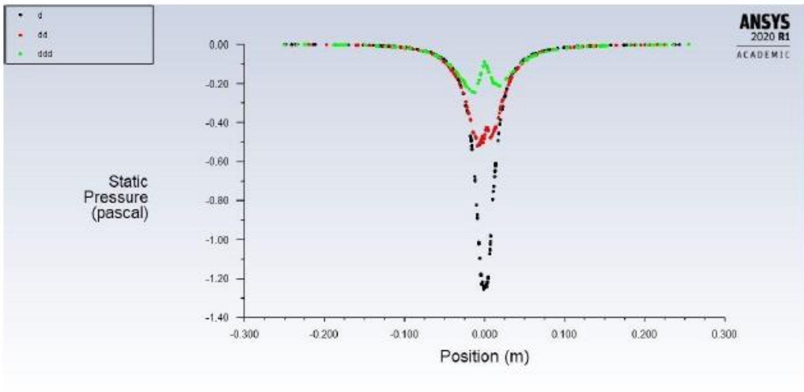


Figure 14. Pressure plot for D, 2D, and 3D downstream locations.

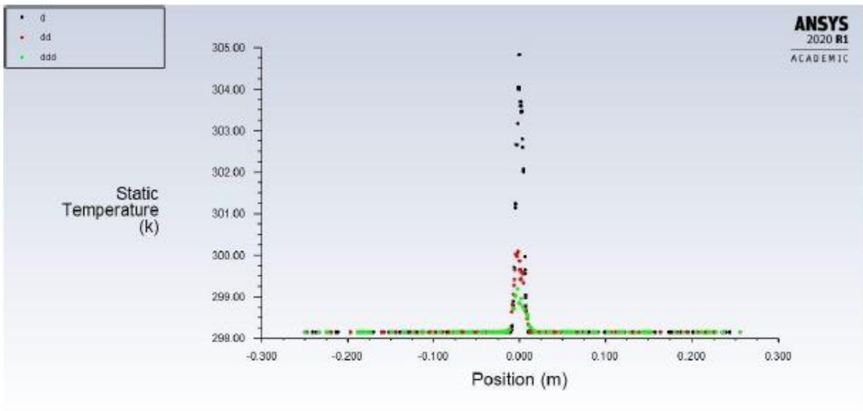


Figure 15. Temperature plot for D, 2D, and 3D downstream locations.

"Flux Report"

| Total Heat Transfer Rate | (w) |
|---|---|
| cylinder_wall | 6.0802173 |
| Net | 6.0802173 |

Figure 16. Flux Report

"Surface Integral Report"

| Area-Weighted Average Surface Heat Transfer Coef. | (w/m2-k) |
|---|---|
| cylinder_wall | 42.67462 |

Figure 17. Surface Integral Report

For part two of this project the following are the contour results for part two:

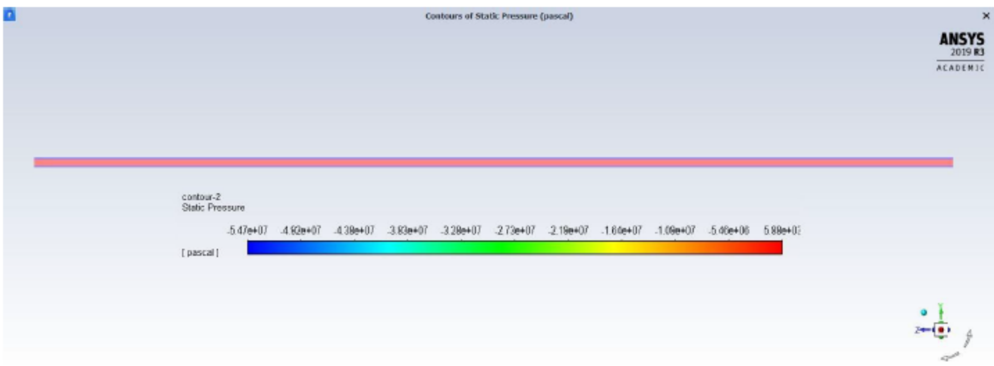


Figure 18. Contours of Static Pressure

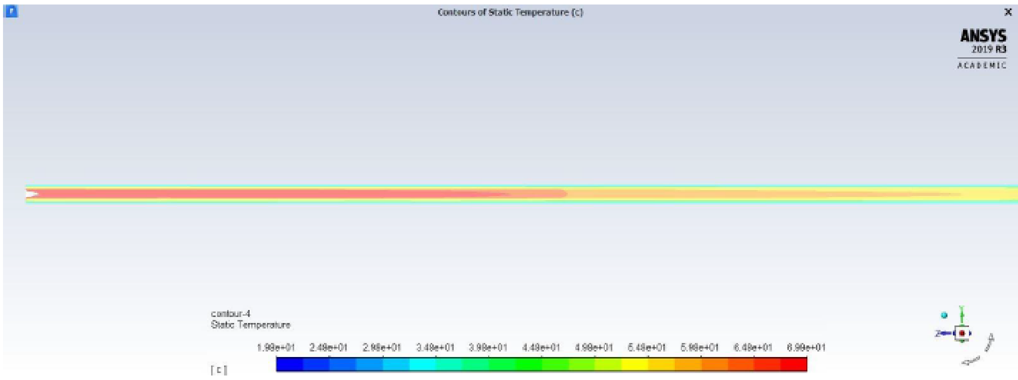


Figure 19. Contours of Static Temperature

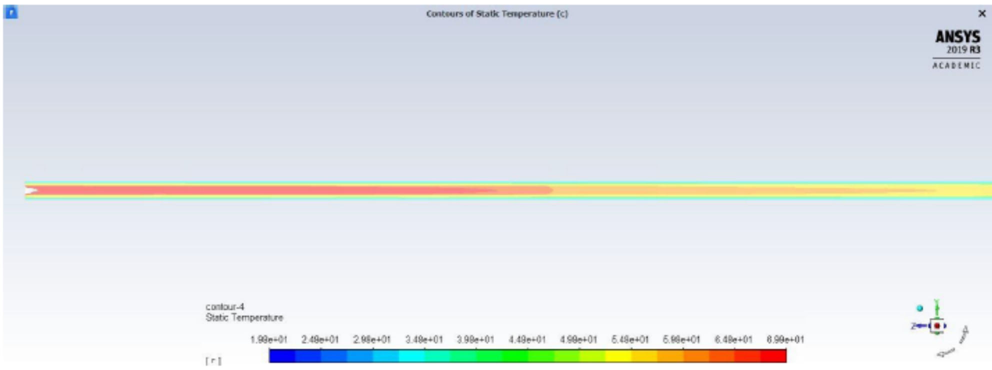


Figure 20. Closer look at Contours of Static Temperature

The following are the resultant plots:

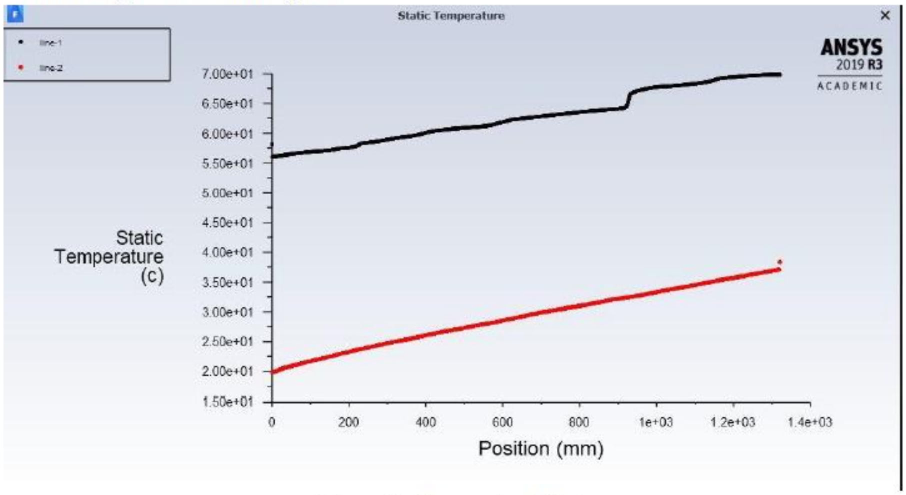


Figure 21. Temperature Plot

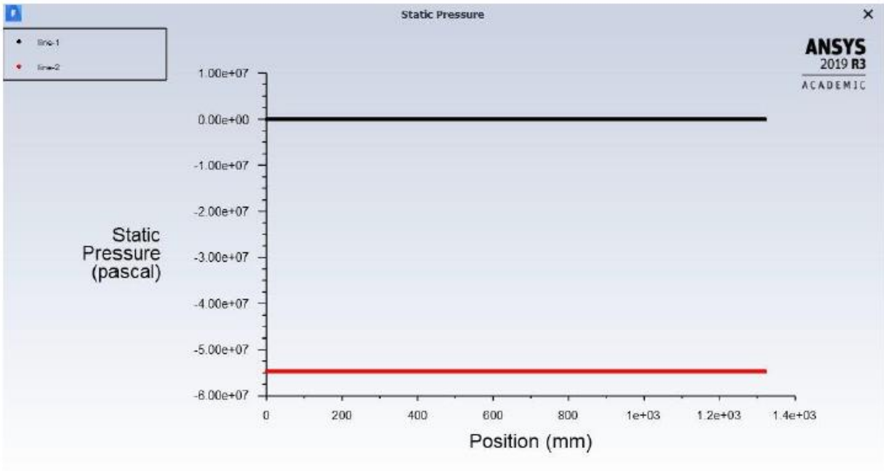


Figure 22. Static Pressure Plot

Using the temperature vs position plot that was given by the Ansys simulation, the NTU, LMTD, and effectiveness can be calculated. Upon close inspection of the plot it can be noted that the hot flow is line 1 (black) and the cold flow line is line 2 (red). Line 1 starts at position 1320 at a temperature of 70C and ends at position 0 with an approximated temperature of 56.5C. Line 2 starts at position 0 with a temperature of 20C and ends at position 1320 with an approximate temperature of 36.5C. Using these temperature values the calculations can be completed

$$\Delta T_{lm} = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1/\Delta T_2)} = \frac{33.5-36.5}{-0.0857666} = 34.97$$

The value of LMTD calculated in the analytical portion was found to be 39.95. From the outlet temperatures given by ansys the LMTD value was found to be 34.97. This difference comes from the difference in outlet temperatures that came from the different methods.

Using the modeled difference in temperatures a new value for actual heat transfer was calculated. Given that all other factors except the outlet temperatures remained the same.

$$q = UA_s\Delta T_{lm} = (599.09*0.0394*34.97) = 825.43 \text{ W}$$

Knowing that everything else stays the same the value of Qmax remains to be 3554.7 W
Using these 2 values the efficiency of the HE modeled in Ansys was determined to be

$$\varepsilon = \frac{Q}{Q_{max}} = \frac{825.43}{3554.7} = 0.232 \text{ or } 23.2\%$$

The value of q that was calculated analytically before the model was 944.1 W and yielded and efficiency of 26.5 %. The slight variation in efficiencies result from the different outlet temperatures. This can be a result of either an error in the original calculations or even an error in the meshing of ansys.

The value of NTU using the new efficiency is calculated by the relationship between εand Cr

$$NTU = \frac{1}{c-1} ln\left(\frac{\varepsilon-1}{\varepsilon C-1}\right) = \frac{1}{0.499-1} ln\left(\frac{0.232-1}{(0.232*0.499)-1}\right) = 0.281$$

The value of NTU that was originally found is 0.3319. The slight difference comes from the different outlet temperatures that were calculated versus what the ansys simulation provided.

The values from the original calculations where outlet temperature was not given are as follow:
U= 599.09 W/m^2*K
NTU=0.3319
ε=0.265
LMTD= 39.5
The work for all these calculations can be found in figures 24&25 in the appendix.

The values using the Outlet temperatures from Ansys are as follow:
U= 599.09 W/m^2*K
NTU=0.281
ε=0.232
LMTD= 34.97

A special note about this water to water HE is that the value of U did not fall in the range of the table that lists the representative values of the overall heat transfer coefficients. The listed range for U is 850-1700. The value calculated for this problem is about 600. This is fine and can be expected in most cases as the values listed within that table is meant to serve only as a guide. A value of U of 600 is not that far off from the min range of 850. Some of the factors that affect the value of U or the fluid velocities , they type of HE, temperatures, and fouling. One of the assumptions made during this problem is that values of water were only considered at the inlets and not at the outlets so an average value for the water properties could not be determined.

## Conclusion

A heated cylinder and heat exchanger were analyzed and also simulated in Fluent Ansys in two different parts. In part one of the project, forced convection was determined by calculating Nusselt number (Nu), Reynold's number (Re), and also heat transfer coefficient (h), for the flow over a cylinder in forced convection. For part one of the project Newton's Law of cooling was also used for a cylinder on forced convection to be able to determine the heat transfer. After finishing the analytical calculations, the simulation using Fluent Ansys was compared to the results obtained into the analytical calculations. Counterflow Heat Exchanger had a similar approach where analytical calculations and simulation using Fluent Ansys had to be performed. The results from the number of transfer units(NTU), effectiveness (E) and the overall heat transfer coefficient (U) from the analytical calculations and from the Fluent Ansys were obtained and realized there was a minimal difference in the final results.

## References:


[1]. Administrator,E.(2020,January31).Whatisthedifferencebetweenparallelflow,counterflow and crossflow heat exchangers?Enerquip.

[2]. https://www.enerquip.com/2018/03/12/whats-the-difference-between-parallel-flow-counter-flow-and-crossflow-heat-exchangers/

[3]. Çengel, Y. A., Cimbala, J. M., & Turner, R. H. (2017). Fundamentals of thermal-fluid sciences. New York, NY: McGraw-Hill Education.

[4]. El Paso, TX Electricity Rates. (2020). Retrieved July 13, 2020, from https://www.electricitylocal.com/states/texas/el-paso/

[5]. Engineering ToolBox, (2003). Convective Heat Transfer. [online] Retrieved June 25, 2020 From https://www.engineeringtoolbox.com/convective-heat-transfer-d_430.html .

[6]. Heat Transfer. (n.d.). Eng Fsu. Retrieved June 25, 2020, from http://www.eng.fsu.edu/%7Eshih/eml3015/lecture%20notes/heat-transfer.htm

[7]. Heat Transfer Mechanisms. (n.d.). Engr Colostate. Retrieved June 25, 2020, from https://www.engr.colostate.edu/%7Eallan/heat_trans/page4/page4f.html#:%7E:text=Convection%20heat%20transfer%20is%20energy,solid%20temperature%20difference%20Ts%2DTf.

[8]. Hire Orlando Workplace Injury Lawyers: Frank M. Eidson P.A. (2020, May 01). Retrieved July 14, 2020, from https://www.frankeidson.com/practice-areas/workers-compensation/workplace-accidents/power-tool-accidents/

[9]. The Editors of Encyclopaedia Britannica. (2019, July 4). heat transfer | Definition & Facts. Encyclopedia Britannica. https://www.britannica.com/science/heat-transfer