MAE 589 - 2022

DESIGN, ANALYSIS AND TRADE-OFF COMPARISON OF HEAT EXCHANGERS PRODUCED BY ADDITIVE MANUFACTURING

Venkateshh Miryalkar[1], Shubhvarata Dutta[1], Leela Krishna Sai Tharun Kammili[1], Abhishek Brahma[1]

[1]Mechanical Engineering Arizona State University, Tempe, AZ, USA

ABSTRACT

In this report we analyze and compare the effectiveness of two different 3D printed Heat Exchanger models. The two models in question are Sine and Straight Channel Core H.E. composed of copper. Copper was chosen as the material due to its high thermal conductivity and water as the flowing fluid for both hot and cold at extremities of the temperature scale. We started off by drafting and modelling the H.E. on SolidWorks followed by a thorough analysis and computation on ANSYS. Certain B.C.'s was defined with the help of Energy Equations and K-ω Turbulence models in the process and assumed for better predicament of the results resulting in obtaining close-form results. On experimenting and analyzing, the results concluded that the effectiveness of the Sine Channel Core H.E. achieved higher rates and pressure drops across the micro-channels as compared to the other Straight Channel Core H.E.

Keywords: Heat Exchanger, Pressure, Temperature, ANSYS.

NOMENCLATURE

Re	Reynold's Number
μ	Dynamic Viscosity
v	Kinetic Viscosity
ρ	Density of the Fluid
L	Characteristic Length
U	Overall Heat Transfer
A_s	Overall Surface Area
δT	Temperature Difference
$T_{h,in}$	Temperature of Hot Fluid @ Inlet
$T_{c,in}$	Temperature of Cold Fluid @ Inlet
$T_{h,out}$	Temperature of Hot Fluid @ Outlet
$T_{c,out}$	Temperature of Cold Fluid @ Outlet
C_p	Specific Heat Capacity
R	Resistance
U_{h}	Overall Heat Transfer co-ef at Hot Fluid Side
U_c	Overall Heat Transfer co-ef at Cold Fluid Side
A_{h}	Contact Surface Area of Hot Fluid Side
A_c	Contact Surface Area of Cold Fluid Side
$\dot{m}_{\mathrm{h}}$	Mass Flow Rate of Hot Fluid
$\dot{m}_c$	Mass Flow Rate of Cold Fluid
Q	Source Term
u	Velocity in X-direction
v	Velocity in Y-direction
w	Velocity in Z-direction

1. INTRODUCTION

Heat Exchangers are devices that allow heat to be transferred between two fluids that are at different temperatures without mixing. Heat Exchangers are utilized in a variety of applications, from residential heating, cooling to industrial chemical processing and power production.[1]

Heat Exchangers of the Shell and tube type offer the highest temperature and pressure drop to minimum. However, these HE of shell and tube type are bulky and huge due to very low surface area to volume ratio, and not highly economical for high-temperature and high-pressure applications. Plate-and-frame Heat Exchangers and plate-fin Heat Exchangers offer higher surface area to volume ratio than shell-and-tube Heat Exchangers, but they are restricted to low or moderate temperature and pressure applications. Besides, pressure drop in those two Heat Exchangers is relatively high and they are prone to fouling due to narrow passages. Printed circuit Heat Exchangers achieve high thermal performance and are small due to the minimal channel geometries. However, they give way for a larger pressure drop due to long straight micro-channels and they are relatively costly due to use of chemical etching process.

3D printing also known as Additive manufacturing, has been a technologically advanced production technology since several decades. Recent developments in Metal additive manufacturing processes in the past few years have opened great new opportunities in this field. Metal additive manufacturing processes like Direct Metal Laser Sintering (DMLS), SLM (Selective Laser Melting), etc. use a high power-density laser beam to melt and fuse metal powders layer by layer to generate finished products. Due to the layer-wise buildup, this process can provide components of virtually any shape and design enabling the manufacturing of more complex geometries and internal structures which were not possible earlier through traditional manufacturing techniques. With the increased design freedom engineers can design more efficient H.E.'s while reducing the number of required assembly parts. This results in H.E.' models with more compact designs which occupy less space, use less material and are lighter in weight that can manage higher thermal loads and power densities. In one of the best results, it is shown that the additive manufacturing Heat Exchanger met the pressure drop and heat transfer design requirement with 66% lower weight and 50% lower volume than the conventional Heat Exchangers. Additionally, additive manufacturing offers a variety of materials such as 17-4 stainless steel, Inconel 625 and 718, aluminum AlSi10Mg, and titanium Ti6Al4V which can be used for different applications. Some of the complex geometries like TPMS (Triply periodic minimal surface) lattice structures which offers a high surface area to volume ratio are only possible to manufacture through additive manufacturing process. Figure shows the design of 2 Heat Exchanger cores with TPMS lattice structure.

As a result, in order to demonstrate the benefits of the additive manufacturing process, we created our own tiny H.E. models and performed numerical analysis on them. We need to improve the accessible surface area of Heat Exchangers in order to build effective Heat Exchangers. It is therefore accomplished using micro-channel pin fins and lattice structures. For our model, we employed micro-channels. Increasing the turbulence of the flow inside the internal channels is another technique to improve Heat Exchanger efficiency. Our second model uses a sinusoidal curve as channel geometry, after that, SolidWorks was used to create 3-D models, which were then imported into Ansys. The resulting simulations are then utilized to do the numerical analysis and acquire the findings. Based on the findings, we've determined which parameter has the greatest influence on heat exchange rate while causing the least amount of pressure drop across, H.E. We can learn a lot about how the additive manufacturing process can help us make better Heat Exchangers via this research.

2. METHODOLOGY

The CFD (Computational fluid Dynamic) analysis in general comprises of three simulation activities: pre-processing, processing, and postprocessing. Figs. 1 represents a schematic diagram of the CFD process applied in this simulation work.

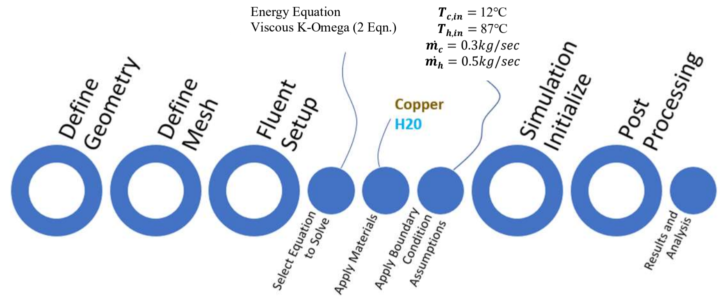

FIGURE 1: ACTIVITIES INVOLVED IN A CFD PROCESS

It begins with the pre-processing where the Heat Exchanger geometry is created in a design tool such as SolidWorks, and the created model is imported in ANSYS and further modeled by defining and generating its geometry and corresponding computational meshing. Post meshing is followed by the processing where the set of governing equations, simulation numerical model, materials, and the boundary conditions and assumptions for the scenario are defined in ANSYS Fluent. The solution calculations are initiated based on the defined set-up in the Simulation Initialization using a method that best suits the scenario. These calculated results such as the temperature and pressure at the exits of the fluid domain can be automatically viewed and gathered at the post-processing stage. The three simulation activities are thereafter repeated for 'N' number of scenario's providing necessary results considered for the trade-off comparison study.

2.1 Pre-processing

Geometry.

For the project, we started with a 50x50x50 mm cube as a fixed control volume for our H.E. cores. From within this cube,

Two models were created for the project, each with a distinct geometry: 1) Straight Cylindrical channels as shown in figs.3 and 2) Sinusoidal Curved Channels as shown in Figs.2. The hot and cold fluid channels were created in such a way that they could flow in both directions. For both models, the distance between neighboring channels, channel diameters (for Hot and Cold Fluids) were all maintained the same. In addition, the spacing between channel layers (Hot and Cold in cross-direction) was limited to a bare minimum. This step was taken to ensure that the greatest number of channels could be accommodated. Based on the geometrical properties listed in Table. 1.

Parameter	Straight Cylindrical Channels	Sin Curve Channels
Geometry Characteristics	Hot branch/ Cold branch	
Length (L)	50 mm	50 mm
Width(W)	50 mm	50 mm
Height(H)	50 mm	50 mm
Channel Shape	Straight	= sin (0.5 * x)
Channel Length	45 mm	45 mm
Distance b/w two channels	0.25 mm	0.25 mm
Channel Diameter (D)	2 mm	2 mm
Number of Layers (N)	9/9	5/5
Number of Channels	153/136	90/90
Total Heat Transfer Area	0.046298 m^2/ 0.041598 m^2	0.032863 m^2/ 0.032863 m^2
Distance between layers (e)	0.25 mm	3.1 mm
Dist. b/w center of channels (p)	2.25 mm	2.25 mm

TABLE 1: GEOMETRICAL PROPERTIES

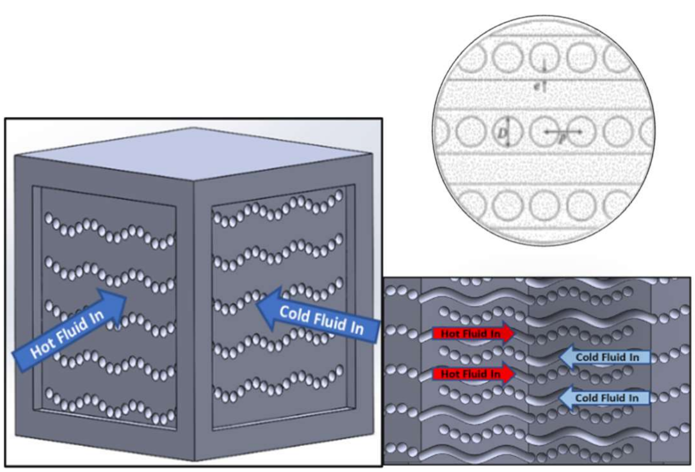

FIGURE 2: SINUSOIDAL CHANNEL

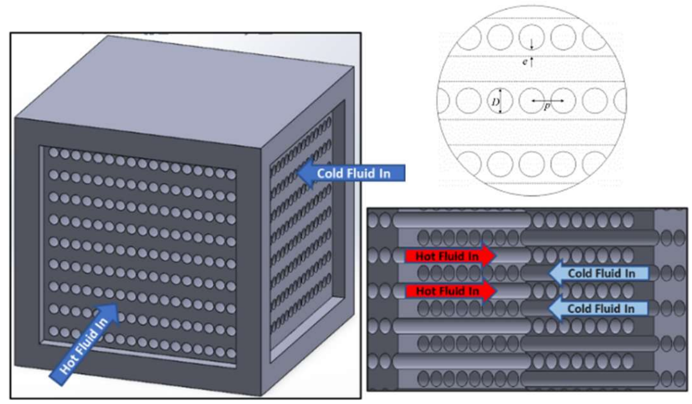

FIGURE 3: STRAIGHT CHANNEL

Feature Creation.

Heat Exchanger (H.E.) model once created in Solid Works, are to be exported as '.igs' files. One advantage of using ANSYS is it supports '.igs' format to import into ANSYS space and allows to claim module in the ANSYS workbench. A solidify feature is initiated to check for stich, gaps and missing faces. Once the solid body of the H.E. model is created, the volume extraction of the fluid flow sections of the H.E. is to be done as shown in Figs. 4.

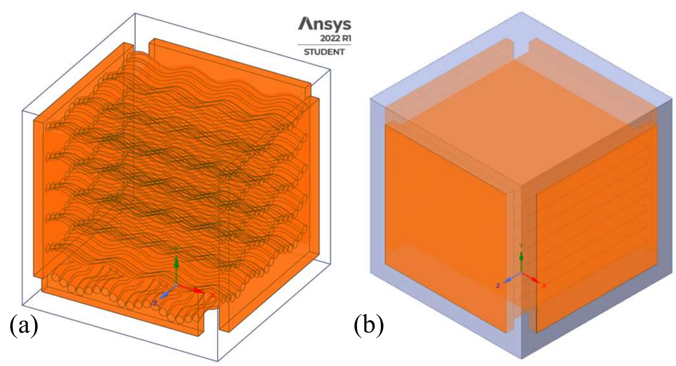

FIGURE 4: VOLUME EXTRACT OF HOT AND COLD FLUID DOMAINS OF: (A) SINE CORE HEAT EXCHANGER (B) MESHING VIEW OF STRAIGHT CORE HEAT EXCHANGER

Computational Meshing.

Before the starting the analysis, the first step was to initiate the Finite Element modelling in the ANSYS mesh in the workbench. Meshing diagrams along with the mesh section of H.E. are shown in Figs. 5.

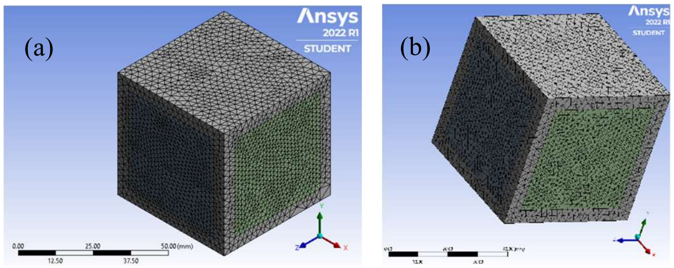

FIGURE 5: (a) MESHING VIEW OF SINE CORE HEAT EXCHANGER (b) MESHING VIEW OF STRAIGHT CORE HEAT EXCHANGER

meshing as shown above was done using ANSYS's default meshing feature. The elements of the mesh for the H.E. models are a mixture of both the tetrahedral and hexahedral cells. These serve as the feature set points for the temperature and flow of the fluids during the numerical simulation in the fluid domains of the hot and cold fluid volumes. Face and Body 'Name selection' was used to define the areas of hot fluid domain, cold fluid domain and the solid H.E. Body.

TABLE 2: MESHING CHARACTERISTICS

Characteristic	Sine Core Heat Exchanger	Straight Core Heat Exchanger
Solver	Fluent	Fluent
Element Order	Linear	Linear
Nodes	105112	130475
Elements	421806	475633

As of the ANSYS Student limitation of the number of mesh cells that can be analyzed in the ANSYS Fluent being less than 500000 cells, a reduction the number of cells was considered. Meshing characteristics of the H.E. models under consideration are given in Table 1, above.

2.2 Processing Setup

Governing Equations.

1. Reynolds Number: In fluid flow scenarios, the Reynolds number (Re) aids in the prediction of flow patterns. Turbulence in a fluid flow is caused by changes in the speed and direction of the fluid, which can travel in the opposite direction of the flow's main direction.
Reynolds number which is calculated as a ratio of inertial forces to viscous forces:

$$Re = \frac{uL}{\nu} = \frac{\rho uL}{\mu}$$

..(1)

Turbulent flow occurs at high Reynolds numbers and is dominated by inertial forces, which tend to produce chaotic eddies, vortices, and other flow instabilities. Laminar flow occurs at low Reynolds numbers, where viscous forces are dominant, and is characterized by smooth, constant fluid motion; turbulent flow occurs at low Reynolds numbers and is dominated by inertial forces, which tend to produce chaotic eddies, vortices, and other flow instabilities.

2. Heat transfer: In a system with a mass flow rate of m kg/sec and a specific heat transfer coefficient of c_p, as well as the resulting change in body temperature dT. The system's heat transport is examined as:

$$(\dot{Q}) = \dot{m}_c c_{pc}(T_{c,out} - T_{c,in}) \text{ (Watts)}$$

..(2)

$$(\dot{Q}) = \dot{m}_{\mathrm{h}} c_{\mathrm{ph}}\left(T_{\mathrm{h},in} - T_{\mathrm{h},out}\right) \text{ (Watts)}$$

..(3)

Heat transfer from cold and hot body is given above.

Heat Capacity: When analyzing Heat Exchangers, it's common to combine the product of the mass flow rate and the fluid's specific heat into a single number known as Heat capacity. The heat capacity rate is a quantity that is defined for both hot and cold fluid streams as:

$$C_c = mC_{pc} \text{ or } C_{\mathrm{h}} = mC_{p\mathrm{h}}$$

..(4)

Minimum heat capacity coefficient is the minimum of both hot and cold heat capacities.

3. Max Heat transfer: It is the maximum heat transfer that can occur within the system, i.e., for a Heat Exchanger max heat transfer can occur when the hot fluid is cooled down to cold fluid's temperature or vice versa.in both the cases, Max heat transfer is calculated as, for a H.E.:

$$(\dot{Q}_{max}) = C_{min}\left(T_{\mathrm{h},in} - T_{\mathrm{h},out}\right) \text{ (Watts)}$$

..(5)

Where C_{min} Is the minimum heat capacity

4. Effectiveness: The effectiveness of a Heat Exchanger allows us to calculate the heat transfer rate without knowing the fluids' output temperatures.

The effectiveness of a Heat Exchanger is determined by the Heat Exchanger's shape as well as the flow arrangement. It is calculated as ratio of heat transfer that is observed to the maximum heat transfer that can be achieved with in the Heat Exchanger:

$$(\xi) = \frac{\dot{Q}}{\dot{Q}_{max}}$$

..(6)

5. LMTD: The temperature difference between hot and cold fluids changes along the Heat Exchanger, therefore having a mean temperature difference is helpful and defined as:

$$(\Delta T_{lm,CF}) = \frac{\Delta T_1 - \Delta T_2}{\ln\left(\Delta T_1 / \Delta T_2\right)} \quad ..(7)$$

$$\Delta T_1 = T_{h,in}(°C) - T_{c,out}(°C) \quad ..(8)$$

$$\Delta T_2 = T_{h,out}(°C) - T_{c,in}(°C) \quad ..(9)$$

Here, ΔT_1 *and* ΔT_2 are cross flow temperature differences within the Heat Exchanger for both hot and cold fluids.

6. Overall heat transfer coefficient: It is advantageous to combine all the thermal resistances in the path of heat flow from the hot fluid to the cold fluid into a single resistance R in the study of Heat Exchangers, and to represent the rate of heat transfer between the two fluids as:

$$\dot{Q} = \frac{\delta T}{R} = UA_s\delta T = U_h A_h \delta T = U_c A_c \delta T \quad ..(10)$$

Numerical Modelling.

1. **K - Omega Turbulence Model**: The equation of turbulence model used in the estimation of the Reynolds-Averaged Navier–Stokes equations in CFD (Computational Fluid Dynamics). This model utilises two partial differential equations for two variables, k and, with the first variable being the Turbulence Kinetic Energy (k) and the second (ω) being the rate of dissipation, to try to anticipate turbulence (of the Turbulence Kinetic Energy (k) into internal Thermal Energy.[3]

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \rho P - \beta\rho\omega k + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\sigma_k(\rho k)}{\omega}\right)\frac{\partial k}{\partial x_j}\right] \quad ..(11)$$

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \left(\frac{\alpha\omega}{k}\right)P - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\sigma_\omega(\rho k)}{\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \frac{\rho\sigma_d}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \quad ..(12)$$

2. **SIMPLEC:** The SIMPLEC (Semi-Implicit Method for Pressure Linked Equations-Consistent) technique is a regularly used numerical procedure in Computational Fluid Dynamics for solving the well-known Navier–Stokes equations.

   The algorithm is designed to be iterative. p*, u*, and v* are all guesses. p', u', v' are the correction terms, and p, u, v are the correct fields, Φ is the property for which we are solving, and 'd' terms are involved with the under-relaxation factor.

3. **Second Order Upwind:** To solve hyperbolic differential equations, we use the second order upwind approach. For the approximation of spatial derivatives, we use three data points that provide a more accurate finite differential stencil. For best results, the three data points we analyze should be continuous and in the same direction.

4. **Quick:** Quick method in ANSYS is used to solve momentum equations with pressure terms. The QUICK method in ANSYS Fluent may be used to compute a higher-order value of the convection variable φ (phi) at a face. The weighted average of second order-upwind and central interpolations of the variable is used in QUICK-type systems.

5. ***Energy Equations model***: Equations That are solved in ANSYS by different methods:

Continuity Equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad ..(13)$$

X-Momentum Equation:

$$\frac{\partial u}{\partial t} + \frac{\partial u^2}{\partial x} + \frac{\partial uv}{\partial y} + \frac{\partial uw}{\partial z} = \frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) + Q_u(x,y,z,t) \quad ..(14)$$

Y-Momentum Equation:

$$\frac{\partial v}{\partial t} + \frac{\partial uv}{\partial x} + \frac{\partial v^2}{\partial y} + \frac{\partial vw}{\partial z} = \frac{1}{Re}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) + Q_u(x,y,z,t) \quad ..(15)$$

Z-Momentum Equation:

$$\frac{\partial w}{\partial t} + \frac{\partial uw}{\partial x} + \frac{\partial vw}{\partial y} + \frac{\partial w^2}{\partial z} = \frac{1}{Re}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) + Q_u(x,y,z,t) \quad ..(16)$$

Energy Equation:

$$\rho c_p\left(\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) \quad ..(17)$$

Boundary Conditions.

For simulation within the Heat Exchangers, Sine Core and Straight Core flow purpose, the following boundary condition assumption were made:

TABLE 2: BOUNDARY CONDITIONS

Properties	Sine Core Heat Exchanger	Straight Core Heat Exchanger
H.E. Material	Copper	Copper
Inlet Fluid	Water	Water
Outlet Fluid	Water	Water
$T_{h,in}$	87°C	87°C
$T_{c,in}$	12°C	12°C
$\dot{m}_h$	0.3kg/sec	0.3kg/sec
$\dot{m}_c$	0.5kg/sec	0.5kg/sec

2.3 Analysis and Post Processing

Once the governing equations, numerical models and the boundary conditions was setup. The analysis of the H.E. was done in ANSYS Fluent Student Edition. The solutions were processed with the SIMPLEC method, where the evaluation of the results was performed from all the regions with one thousand iterations until the solution converged.

One of the numerical models was solved for the given domain and the conditions setup. It is the post processing that regulates the output from the ANSYS simulations. ANSYS provides automatic reports for the weighted surface area average of the outlet temperature and outlet pressure at the inlet and exit of the H.E. after calculating the solutions. The total heat transfer rate, outlet temperatures and pressure drop of the hot and cold fluid sections of the H.E. were derived from formulae pertaining to Section 2.2 Governing Equation.

3. RESULTS AND DISCUSSION

This section deals with the results of thermal analysis conducted on the H.E.s with Sine curve and Straight Channel Core.

Temperature Distribution Analysis.

Volume renderings of temperature were generated for the whole volume of the H.E. models i.e., Sine and Straight to emphasize on the temperature distribution in the H.E. as a result of steady flow of water in the hot and cold fluid domain. Figs. 3 and Figs. 4 shows the differences between the volumetric temperature variance of water across the volume of the H.E. The renderings display the thermal behavior of the fluid in the inlet (covered in red shade) and outlet regions (covered in lighter shade per scale), and on surfaces before, after and in between the channels. As expected, the fluid temperature is uniform at the inlet region of the fluid regions in the models. A difference in temperature initially is evident in the cross-sections of both hot fluid and the cold fluid of the H.E.'s.

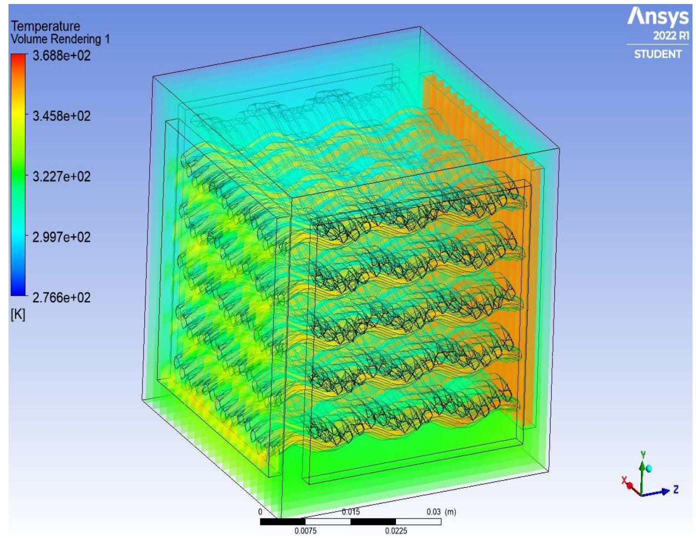

FIGURE 3: TEMPERATURE CONTOUR OF SINE CORE HEAT EXCHANGER

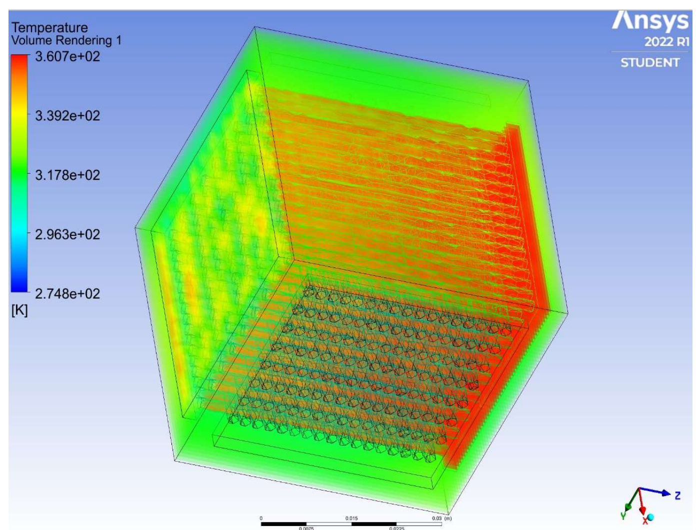

FIGURE 4: TEMPERATURE CONTOUR OF STRAIGHT CORE HEAT EXCHANGER

The difference in temperature in the hot fluid domain when compared is more prominent, such that the upper inlet portion of the H.E. volume is red in color as compared to the coolant side of the H.E. which is greenish blue in color. The temperature gradient observed in the hot fluid domain from the hotter inlet to the colder outlet is in the range of 87°C to 43°C for the Sine Core

H.E. and in the range of 87°C to 53°C in the Straight Core H.E. As a result of complex periodic path travelled by the fluid in the Sine Core H.E. there is more timeframe for the thermal interaction with the cold fluid domain which in turn resulted in the increased temperature gradient in the Sine Core H.E.

Pressure Drop Analysis.

Similar volumetric pressure variation contours for the Sine and Straight Core H.E. were generated to analyze and visualize the pressure drop usually observed in the micro channel H.E. The volumetric pressure contours of Sine and Straight Core H.E. are given by Figs. 5 and 6 respectively.

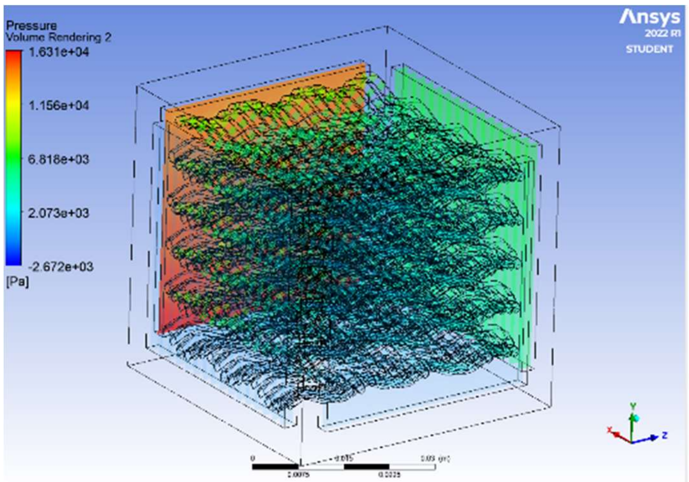

FIGURE 5: PRESSURE VARIATION IN SINE CORE HEAT EXCHANGER

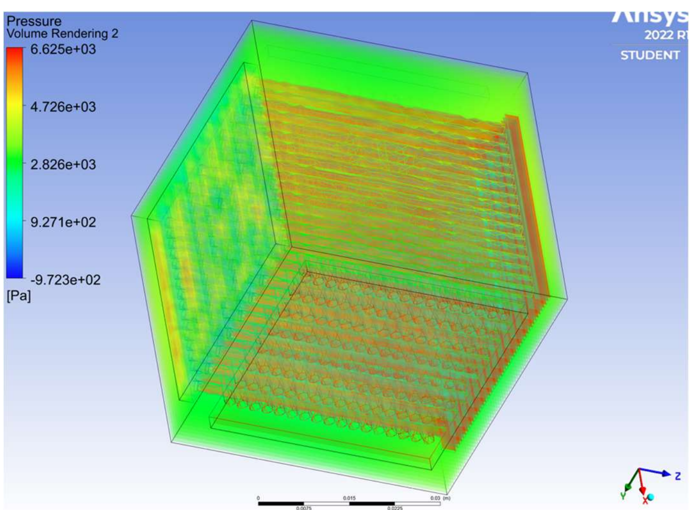

FIGURE 6: PRESSURE VARIATION IN STRAIGHT CORE HEAT EXCHANGER

There is a no table pressure difference present between the inlet and outlet sections of the hot and the cold fluid domains of the Sine and the Straight Core H.E. When compared to the straight Core, the sine Core experienced more pressure loss across the ends. Thus, increased complexity of the traverse profile of the fluid element greater is the pressure loss.

Combined Analysis Results for Trade-Off.

Analysis parameters of a H.E. such as the achieved (LMTD) Log Mean Temperature Difference, outlet pressure, overall Heat Transfer rate, effectiveness and Heat Transfer rate per unit volume are evaluated for the defined scenario for both Sine and Straight Core H.E. The final values are compared in Table. 3, below.

TABLE 3: RESULT CHARACTERISTICS

Parameters	**Sine Core Heat Exchanger**	**Straight Core Heat Exchanger**
Result Characteristics	Hot Fluid / Cold Fluid	Hot Fluid / Cold Fluid
Inlet Temperature	87°C / 12°C	87°C / 12°C
Outlet Temperature	43°C/ 23°C	53°C / 22°C
Log Mean Temperature Difference	-228°C	-221°C
Inlet Pressure	6294.575 Pa/ 15233.66 Pa	1930.55 Pa / 6195.75 Pa
Outlet Pressure	123.64 Pa / 371.19 Pa	42.60 Pa / 136.65 Pa
Heat Transfer Rate	0.038 KW / 0.143 KW	0.027 KW / 0.0999 KW
Effectiveness	0.5772	0.4488
Heat Transfer Rate/Unit Volume	0.43427 MW/m^3	0.337 MW/m^3

As described above the Sine Core H.E. comes with an enhanced Heat Transfer rate per unit volume at 0.43427 MW/m^3 when compared with 0.337 MW/m^3 of Straight Core H.E.

4. CONCLUSION

Model with Straight Core H.E. showed an effectiveness of 44.88%, with pressure variation as 97.8 % and temperature variation as 9.4 % in hot section and 3.5 % in cold section. Whereas the model with Sin (0.5*x) º Channel Core H.E. showed an effectiveness of 57.72 %, with pressure variation as 98.03 % and temperature variation as 12.2 % in hot section and 4.04 % in cold section.

For the same volume effectiveness of Sine Channeled Core H.E. is higher than the Straight Channeled Core H.E.

REFERENCES

[1] Cengel Y. A., Ghajar A. J. "*Heat and Mass Transfer*"*6th edition*.
[2] Peng, Hao; Gao, Feng; and Hu, Wenjing. "*Design, Modeling and Characterization of Triply Periodic Minimal Surface Heat Exchangers with Additive Manufacturing*"
[3] Cruz, P.A.D., et al. "*Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nanofluid in a shell and tube heat exchanger*"
[4] Pal, D., et al. "*Temperature distribution analysis of Cu, Al and Steel material heat exchangers by ANSYS*"
[5] *ANSYS FLUENT 12.0 User's Guide* https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node366.htm.
[6] W. Rodi, N. Fueyo., et al. "*Engineering Turbulence Modelling and Experiments 5*"
[7] INCROPERA, F. P.; DEWITT D. P.; et al. "*Fundamentals-of-heat-and-mass-transfer-6th-edition"*
[8] D.CruzaEd-Jefferson P. A.; Nuquib E.Y.P.E. "*Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nanofluid in a shell and tube heat exchanger,*"