

Cross flow heat exchanger: Tube banks

In many engineering applications in an industry, the fluid flows over the number of tube surfaces so as to exchange the heat between the two fluids. In most of the such cases, the direction of the two fluids is usually orthogonal. The orthogonality in the direction of the two fluids leads to the development of cross flow heat exchanger. The cross flow heat exchanger has wide application in many commercial sector, such as air conditioner coil, economizer, tube boiler, automotive radiator, air heater, chillers.

According to the type of application under investigation, cross flow heat exchangers can have different types in terms of shape and sizes. In this project, we are going to investigate the design of tube banks of these heat exchangers. Fig. 1 shows the overview of this type of heat exchanger. This figure shows the heat exchange inside the evaporator, so that the refrigerant fluid with a constant temperature is turned into saturated vapor and the air passing through the pipes is used for cooling and air conditioning.

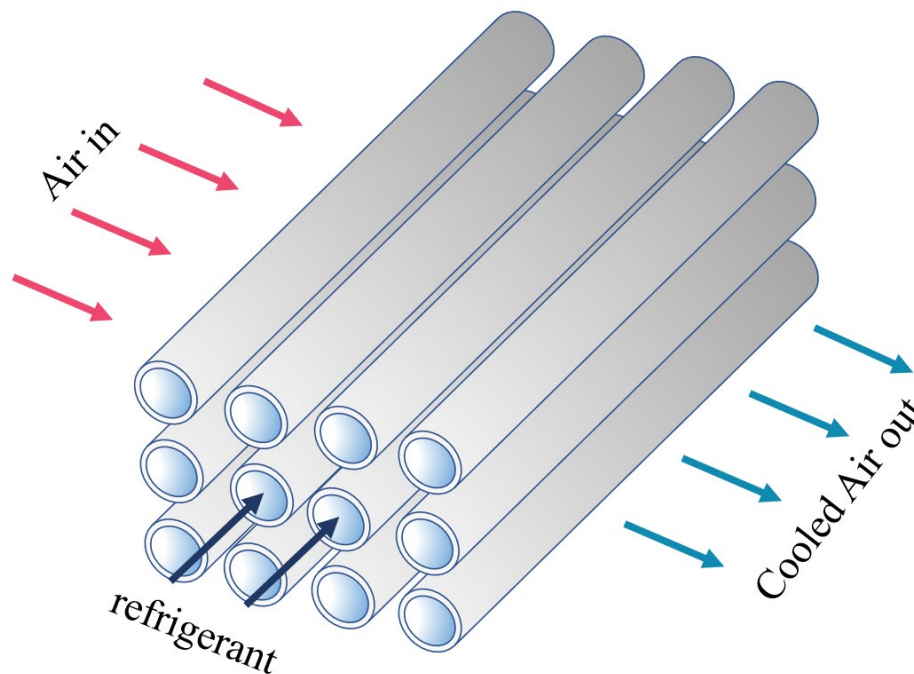


Fig. 1. Flow across banks of tubes

Modeling

To simplify the project, changes in temperature and air velocity in the direction of refrigerant flow are ignored (2D-modeling). The schematic of the desired system is shown in Fig. 2. According to this modeling,

4 rows of tubes are placed staggered in the direction of the fluid velocity in a channel. dimensions and initial conditions required for modeling are given in Table 1.

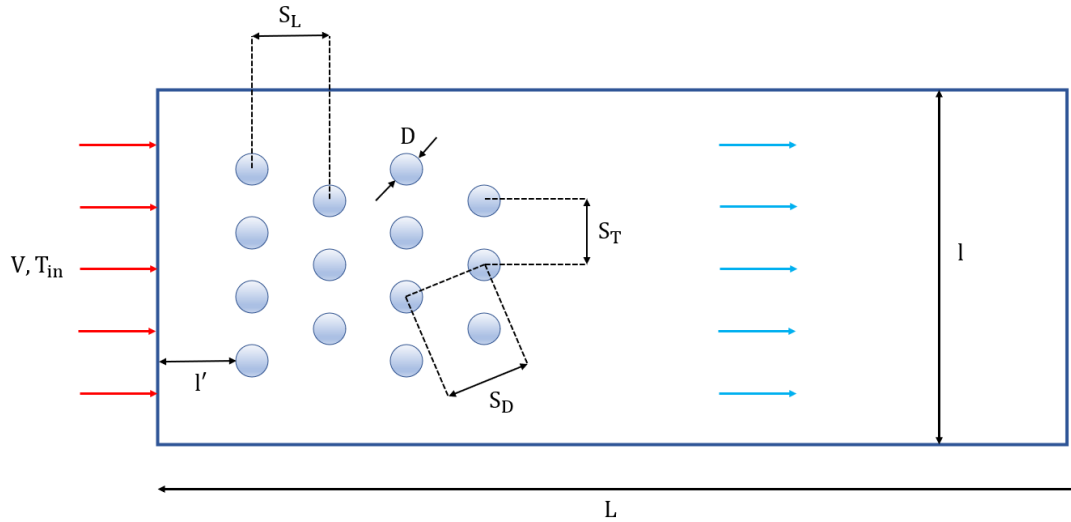


Fig. 2. Schematic of the system

Table 1 Parameters used in modeling

Parameter	Value
D	5 cm
S_L	10 cm
S_T	10 cm
S_D	11.18 cm
l	55 cm
l'	10 cm
L	100 cm
V_{in}	0.5 m/s
T_{in}	300 K
Refrigerant temperature	275 K

Modeling of this system with thermal and hydrodynamic analysis is done using Fluent software. The steps of modeling are summarized below.

- The initial geometry design is done using DesignModeler in Ansys software.
- After obtaining the geometry, in the Ansys Meshing environment, start creating the finite volumes. (During the meshing, make sure to create the boundary layer mesh. The mesh size should also be

selected so that the results are independent of it). Then name the inlet and outlet of the fluid, the walls and, if needed, axisymmetric.

- Enter the obtained meshed geometry into the Fluent environment and solve the problem using appropriate initial and boundary conditions as well as a justifiable solution method.

Assumptions

1. Consider air properties (density, viscosity, heat capacity, etc.) as constant and equal to its value at T_{in} .
2. Ignore the thermal resistance of the refrigerant tubes and consider the temperature of each tube equal to the temperature of the refrigerant.
3. Consider the boundary conditions for each of surface according to Table 2.

Table 2 Boundary conditions

Surface	Boundary condition
Inlet	Velocity inlet
Outlet	Pressure outlet (tick prevent reverse flow)
Upper and lower walls	Far field OR insulation wall
Tubes	Isothermal

4. To simulate problems such as the desired project where there is an obstacle in front of the fluid flow, to simulate the turbulent flow inside the software, k-omega equations are usually used. (Set solver to SST: k- ω)

The general equations solved by the software for the flow simulation are as follows.

- Continuity (1 equation)
- Momentum (2 equations)
- Energy (1 equation)
- k- ω (2 equations)

5. Since the size of the boundary layer are very important for the correct simulation of energy conservation, you can use the following equation to find the thickness of the first boundary layer.

$$\text{First boundary layer thickness} = \sqrt{74} \text{Re}^{-\frac{13}{14}} y^+ D_h \quad (1)$$

$$y^+ \approx 5 \quad (y^+ < 5) \quad (2)$$

(Note: For more information about the process of solving the project, you can use YouTube, CFDOnline, cfd.ninja and...)

Validation

So far, many experimental results have been performed to obtain the existing relations of the average heat transfer coefficient and also the amount of pressure drop in tube banks heat exchangers. To verify the numerical solution, the results obtained by Zukauskas can be used. These equations and the constant coefficients used for them are given below. For more information, refer to the reference book.

$$\overline{Nu}_D = C_2 \left[C_1 Re_{D,\max}^m Pr^{0.36} \left(\frac{Pr}{Pr_s} \right)^{0.25} \right] \quad (3)$$

$$\Delta T_{lm} = \frac{(T_s - T_i) - (T_s - T_o)}{\ln \left(\frac{T_s - T_i}{T_s - T_o} \right)} \quad (4)$$

$$\frac{T_s - T_i}{T_s - T_o} = \exp \left(- \frac{\pi D N \bar{h}}{\rho V N_T S_T C_p} \right) \quad (5)$$

$$q' = N(\bar{h} \pi D \Delta T_{lm}) \quad (6)$$

Table 3 Constants of equation (3)

Conguration	$Re_{D,\max}$	C_1	m
Aligned	$10-10^2$	0.80	0.40
Staggered	$10-10^2$	0.90	0.40
Aligned	10^2-10^3	Approximate as a single (isolated) cylinder	
Staggered	10^2-10^3		
Aligned ($S_T/S_L > 0.7$) ^a	$10^3-2 \times 10^5$	0.27	0.63
Staggered ($S_T/S_L < 2$)	$10^3-2 \times 10^5$	$0.35(S_T/S_L)^{1/5}$	0.60
Staggered ($S_T/S_L > 2$)	$10^3-2 \times 10^5$	0.40	0.60
Aligned	$2 \times 10^5-2 \times 10^6$	0.021	0.84
Staggered	$2 \times 10^5-2 \times 10^6$	0.022	0.84

^aFor $S_T/S_L < 0.7$, heat transfer is inefficient and aligned tubes should not be used.

Table 4 Correction factor C_2

N_L	1	2	3	4	5	7	10	13	16
Aligned	0.70	0.80	0.86	0.90	0.92	0.95	0.97	0.98	0.99
Staggered	0.64	0.76	0.84	0.89	0.92	0.95	0.97	0.98	0.99

Tasks

The main tasks of the project are mentioned below:

- Report the assumed values in the meshing section as well as the assumptions considered for the modeling in fluent.
- Obtain the convergence history for the governing equations of the problem. (Residuals $< 10^{-6}$)
- Report temperature, velocity and pressure contours.
- The amount of heat transfer exchanged in each tube and the total heat transfer should be reported. Compare the amount of heat transfer in the initial tubes and the tubes at the end of the channel. Justify the result obtained.

Extra marks

- 3D modeling of the system
- Compare the modeling results in the software using the experimental equations mentioned in the previous section. If there is a difference between these values, explain the reason for this difference.

To submit the assignment, the following documents are required:

- Word
- PDF
- ANSYS modeling file