International Journal of Scientific Engineering and Applied Science (IJSEAS) – Volume-2, Issue-3,March 2016
ISSN: 2395-3470
www.ijseas.com

Designing and Performance Evaluation of a Shell and Tube Heat Exchanger using Ansys (Computational Fluid Dynamics)

Digvendra singh[1], Narayan Das Pal[2]

[1] Post graduate student, Bansal Institute of Research and Technology, Bhopal, India.

[2] Assistant Professor, Bansal Institute of Research and Technology, Bhopal, India

ABSTRACT

This work is based designing and performance Evaluation of a Shell and Tube Heat Exchanger using Ansys (Computational Fluid Dynamics). Heat exchanger has a variety of applications in different industries and in this study one such heat exchanger is taken into account. The heat exchanger is designed as per the commercial needs of the industry. In current manufacturing process in the steel industries water is being used both as shell and tube side in heat exchanger. The shell side fluid is water exit from the heat exchanger and goes to cooling tower for cooling of fluid but water is not sufficient cooled by the cooling tower for again using in heat exchanger.

The new designing dimension of a shell and tube heat exchanger having 21 tubes, 170mm shell diameter and 610 long in this design does not include the type of the header to be used, so I have taken three types of header for analysis which can provide a uniform velocity in the inlet of each tube. Modifying the heat exchanger the effectiveness of heat exchanger is increased by changing the shell side fluid (from water to methanol). Methanol provide sufficient cooling to as compared water in heat exchanger and increase the effectiveness of heat exchanger.CFD simulations are used for the optimum positioning of the inlet nozzle which could be proposed from the uniform distribution of the liquid methanol and the uniform velocity distribution though each and every tube. The main objective of this research project is to verify the heat exchanger designed with the use of the Kern's technique, by the use of Commercial Computational Fluid Dynamics (CFD) software. For the simulation, purpose a symmetric view of the simplified geometry of the heat exchanger is made using Solidworks software. In the present study, CFD simulation is used to study the temperature and velocity profiles through the tubes and the shell. It is found that the heat transfer through the length of the tube is not uniform.

Keywords: *heat exchanger, kern's technique, CFD, solidworks software*

1. INTRODUCTION

Heat exchangers are devices used to transfer heat energy from one fluid to another. Typical heat exchangers experienced by us in our daily lives include condensers and evaporators used in air conditioning units and refrigerators. Boilers and condensers in thermal power plants are examples of large industrial heat exchangers.

There are heat exchangers in our automobiles in the form of radiators and oil coolers. Heat exchangers are also abundant in chemical and process industries.

Different heat exchangers are named according to their applications. For example, heat exchangers being used to condense are known as condensers; similarly heat exchangers for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transferred using least area of heat transfer and pressure drop. A better presentation of its efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provide an insight about the capital cost and power requirements (Running cost) of a heat exchanger. Usually, there are lots of literature and theories to design a heat exchanger according to the requirements. A good design is referred to a heat exchanger with least possible area and pressure drop to fulfil the heat transfer requirements [1].

1.1 Classification of Heat Exchanger

Heat exchangers are typically classified according to flow arrangement and type of construction. The simplest heat exchanger is one for which the hot and cold fluids move in the same or opposite directions in a concentric tube (or double-pipe) construction. In the parallel flow arrangement of Figure 1.1a, the hot and cold fluids enter at the same end, flow in the same direction, and leave at the same end. In the counter flow arrangement of Figure 1.1b, the fluids enter at opposite ends, flow in opposite directions, and leave at opposite ends. Alternatively, the fluids may move in cross flow (perpendicular to each other), as shown by the finned and without fin tubular heat exchangers of Figure 1.2. The two configurations are typically differentiated by an idealization that treats fluid motion over the tubes as unmixed or mixed. In Figure 1.2a, the fluid is said to be unmixed because the fins inhibit motion in a direction (y) that is transverse to the main-flow direction (x). In this case the fluid temperature varies with x and y. In contrast, to the unfinned tube bundle of two configurations are typically differentiated by an idealization that treats fluid motion over the tubes as unmixed or mixed. In Figure 1.2a, the fluid is said to be unmixed because the fins inhibit motion in a direction (y) that is transverse to the main-flow direction (x). In this case the fluid temperature varies with x and y. In contrast, to the unfinned tube bundle of Figure 1.2b, fluid motion, hence mixing, in the transverse direction is possible, and temperature variations are primarily in the main flow direction. Since the tube flow is unmixed, both fluids are unmixed in the finned exchanger, while one fluid is mixed and the other unmixed in the unfinned exchanger. The nature of the mixing condition can significantly influence heat exchanger performance. Another common configuration is the shell-and-tube heat exchanger. Specific forms differ according to the number of shell-and-tube passes, and the simplest form, involves single pass.

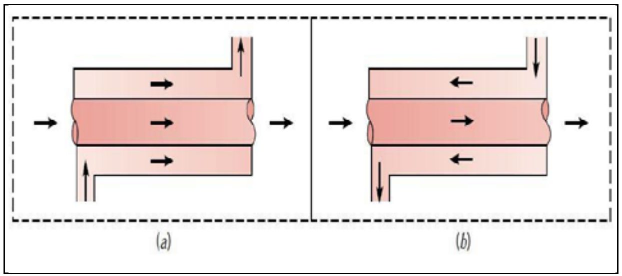


Figure:-1.1 Concentric tube heat exchangers. (a) Parallel flow. (b) Counter flow. [2]

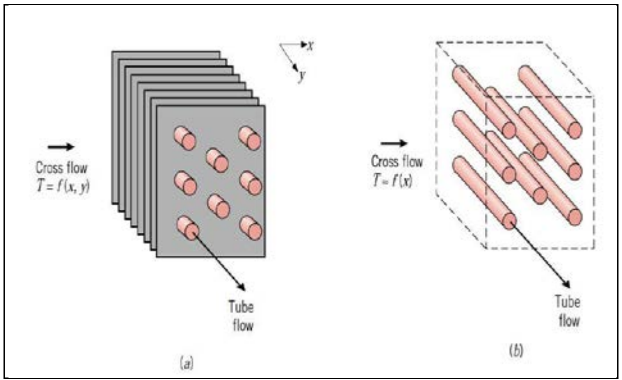


Figure:-1.2 Concentric tube heat exchangers. (*a*) Parallel flow. (*b*) Counter flow. [2]

Baffles are usually installed to increase the convection coefficient of the shell-side fluid by inducing turbulence and a cross-flow velocity component. In addition, the baffles physically support the tubes, reducing flow-induced tube vibration. Baffled heat exchangers with one shell pass and two tubes passes and with two shell passes and four tube passes are shown in Figures 1.4*a* and 1.4*b*, respectively. A special and important class of heat exchangers is used to achieve a very large (≥ 400 m^2/m^3 for liquids and ≥ 700 m^2/m^3 for gases) heat transfer surface area per unit volume. Termed *compact heat exchangers*, these devices have dense arrays of finned tubes or plates and are typically used when at least one of the fluids is a gas, and is hence characterized by a small convection coefficient. The tubes may be *flat* or *circular*, as in Figures 1.5*a* and 1.5*b*, *c*, respectively, and the fins may be *plate* or *circular*, as in Figures 1.5*a*, *b* and 1.5*c*, respectively. Parallel-plate heat exchangers may be finned or corrugated and may be used in single-pass (Figure 1.5*d)* or multipass (Figure 1.5*e*) modes of operation. Flow passages associated with compact heat exchangers are typically small, and the flow is usually laminar.

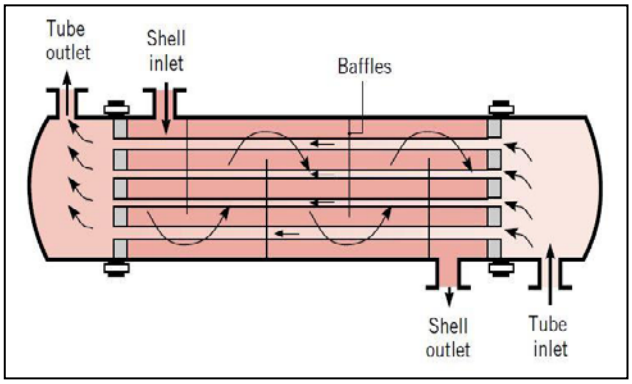


Figure-1.3 Shell-and-tube heat exchanger with one shell pass and one tube pass (cross-counter flow mode of operation) [2]

International Journal of Scientific Engineering and Applied Science (IJSEAS) – Volume-2, Issue-3,March 2016
ISSN: 2395-3470
www.ijseas.com

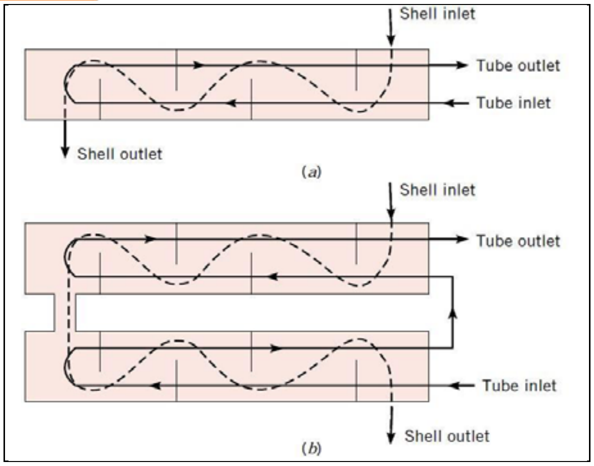


Figure-1.4 Shell-and-tube heat exchangers. (*a*) One shell pass and two tube passes. (*b*) Two shell passes and four tube passes. [2]

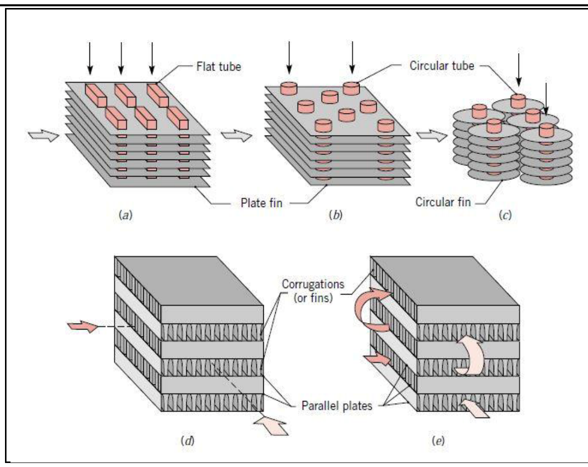


Figure 1.5 Compact heat exchanger cores. (*a*) Fin–tube (flat tubes, continuous plate fins). (*b*) Fin–tube (circular tubes, continuous plate fins). (*c*) Fin–tube (circular tubes, circular fins). (*d*) Plate–fin (single pass). (*e*) Plate–fin (multipass). [2]

1.2 Applications of Heat exchangers

Applications of heat exchangers is a very vast topic and would require a separate thorough study to cover each aspect. Among the common applications is their use in process industry, mechanical equipment's industry and home appliances. Heat exchangers can be found employed for heating district systems, largely being used now days. Air conditioners and refrigerators also install the heat exchangers to condense or evaporate the fluid. Moreover, these are also being used in milk processing units for the sake of pasteurization. The more detailed applications of the heat exchangers can be found in Table 1.1 with respect to different industries [3].

2. PROBLEM IDENTIFICATIONS

In current manufacturing process in the steel industries water is being used both as shell and tube side in heat exchanger. The shell side fluid is water exit from the heat exchanger and goes to cooling tower for cooling of fluid but water is not sufficient cooled by the cooling tower for again using in heat exchanger. After modifying the heat exchanger the effectiveness of heat exchanger is increased by changing the shell side fluid (from water to methanol). Methanol provides sufficient cooling to as compared water in heat exchanger. A Shell and Tube Heat Exchanger to coolant Methanol from 298 K to 331.22 K. Flow-rate of Methanol is 0.055036 kg/sec. Water will be used as the tube side, with a temperature from 373 K to 354.31 K. In current paper, a shell and tube heat exchanger is selected from steel industries. The proposed design will be based on the Kern's method. The design obtained from the Kern's technique is analysed and evaluated using the CFD software which will give us a detailed view of the temperature, velocity and pressure profile. These profiles will have us understand the flow inside the shell and as well as tubes. The flow across the baffle plates will also matter as they provide the turbulence according to the tubes and helps in better heat transfer.

3. METHODOLOGY

3.1 Kern's technique of Shell-And-Tube Heat Exchanger

Kern's technique was based on real industrial work on commercial heat exchanger.

3.1.1 Shell and Tube Heat Exchanger

The number of tubes and size of tubes in heat exchanger depends up on the

- Fluid flow rates.
- Pressure drop.
- The number of tubes and size of tubes selected.
- The lower velocity limit corresponds to limiting the **fouling factor**.
- The upper velocity limit corresponds to limiting the **rate of erosion**.

3.1.2 Tube Layout

- Triangular pitch (30^o layout) is better for heat transfer and surface area per unit length (greatest tube density.)
- Square pitch (45° & 90° layouts) is needed for cleaning.
- The 30°, 45° and 60° are staggered, and 90° is in line.
- For the identical tube pitch and flow rates, the tube layouts in decreasing order of shell-side heat transfer coefficient and pressure drop are: 30°, 45°, 60°, 90°.
- The 90° layout will have the low heat transfer coefficient and the low pressure drop.
- The square pitch (90° or 45°) is used when jet or mechanical cleaning is necessary on the shell side.

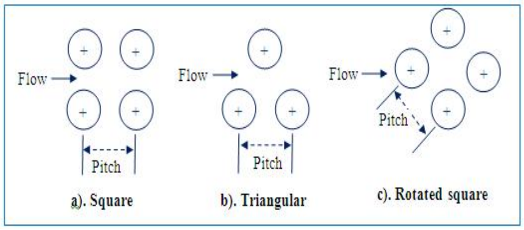


Figure 3.1 Tube Layout and Flow Scales [4]

3.2 Basic Design Procedure and Theory

As the objective of this present project is to evaluate the Kern's technique which is the best technique and by far the most used method of the shell and tube heat exchanger design, below I have mentioned some of the most useful equations for the designing of the shell and tube heat exchanger using Kern's technique.

The general equation for heat transfer across a surface is:

$$\Longrightarrow \qquad Q = UA\Delta T_m \quad [1] \quad \ldots\ldots\ldots\ldots\ldots(4.1)$$

Where,

Q = Heat transferred per unit time, W,

U = the overall heat transfer coefficient, W/m²°C,

A = Heat-transfer area, m²

ΔT_m = The mean temperature difference, the temperature driving force, °C.

Collect physical properties and Heat Exchanger specifications:

Physical properties

Table 3.1 Physical properties of fluids (Methanol & Water)

Physical properties	Methanol Cold fluid	Water Hot fluid
Density (kg/m³)	**785**	**998.2**
Specific heat C_p (J/kg-K)	**2534**	**4182**

Heat Exchanger specifications

- Sea water is corrosive, so assign to tube-side.
- Use one shell pass and two tube passes.
- At shell side, fluid (methanol) is relatively clean. So, use **1.25 triangular pitch.**

  (Pitch: Distance between tube centres)

The prime objective in the design of an exchanger is to determine the surface area required for the specified duty (rate of heat transfer) using the temperature differences available. The overall coefficient is the reciprocal of the overall resistance to heat transfer, which is the sum of several individual resistances. For heat exchange across a typical heat exchanger tube the relationship between the overall coefficient and the individual coefficients, which are the reciprocals of the individual resistances, is given by:

⟹

$$\frac{1}{U_o} = \frac{1}{h_s} + \frac{1}{h_o d} + \frac{d_o \ln \frac{dn}{di}}{2k_w} + \frac{do}{di} \times \frac{1}{h_i} + \frac{do}{di} \times \frac{1}{h_i}$$

[1] (4.2)

Where,

U_o = the overall coefficient based on the outside area of the tube, W/m² ºC,

h_o = outside fluid film coefficient, W/m² ºC,

h_i = inside fluid film coefficient, W/m² ºC,

$h_o d$ = outside dirt coefficient (fouling factor), W/m² ºC,

$h_i d$ = inside dirt coefficient, W/m² ºC,

k_w= thermal conductivity of the tube wall material, W/m ºC,

d_i= tube inside diameter, m,

d_o= tube outside diameter, m.

The magnitude of the individual coefficients will depend on the nature of the heat transfer process (conduction, convection, condensation, boiling or radiation), on the physical properties of the fluids, on the fluid flow-rates, and on the physical arrangement of the heat-transfer surface. As the physical layout of the exchanger cannot be determined until the area is known the design of an exchanger is of necessity a trial and error procedure. The steps in a typical design procedure are given below:

1. Calculate the area for cross-flow As for the hypothetical row of tubes at the shell equator, given by:

$$\Rightarrow A_s = \frac{(P_t - d_o) D_s l_s}{P_t} \quad [1] \qquad (4.3)$$

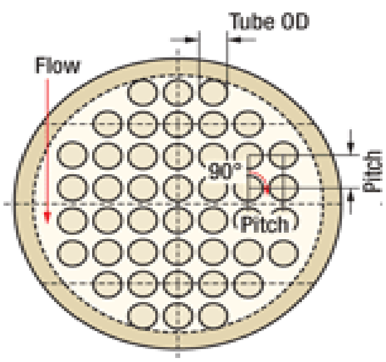


Fig.3.2 Arrangement of Tubes in Heat Exchanger [4]

Where,

P_t = tube pitch,

d_o = tube outside diameter,

D_s = shell inside diameter, m,

l_s = baffle spacing, m.

The term $\frac{(P_t - d_o)}{P_t}$ is the ratio of the clearance between tubes and the total distance between tube centres.

2. Calculate the shell-side mass velocity Gs and the linear velocity us:

$$\Rightarrow \quad Gs = Ws/As \qquad (4.4)$$

$$\Rightarrow \quad us = Gs/\rho \qquad (4.5)$$

Where,

Ws= fluid flow-rate on the shell-side, kg/s,
ρ= shell-side fluid density, kg/m3.

3. Calculate the shell-side equivalent diameter (hydraulic diameter), Figure 1. For a square pitch arrangement:

$$d_e = \frac{1.27}{d_o}(p_t^2 - 0.785 d_0^2) \quad [1]$$

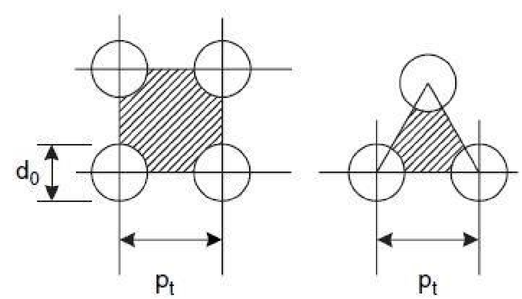


Figure 3.3 Equivalent diameter, cross-sectional areas and wetted perimeters [5]

4. Calculate the shell-side Reynolds number, given by:

$$\mathrm{Re} = \frac{Gsde}{\mu} \quad [1]$$

5. For the calculated Reynolds number, read the value of jh from Figure 2 for the selected baffle cut and tube arrangement, and calculate the shell-side heat transfer coefficient hs from:

$$Nu = \frac{h_s d_e}{k_f} = j_h \,\mathrm{Re}\,\mathrm{Pr}^{1/3}\left(\frac{\mu}{\mu w}\right)^{0.14} \quad [1]$$

Where,

Nu = Nusselt number = $\frac{h_i d_e}{k_f}$

Re = Reynolds number = $\frac{Gsde}{\mu}$

Pr = Prandtl number = $\frac{C_p\mu}{k_f}$

h_i = inside coefficient, W/m² °C,

d_e = equivalent (or hydraulic mean) diameter, m

U_t = fluid velocity, m/s,

k_f = fluid thermal conductivity, W/m °C,

G_s = mass velocity, mass flow per unit area, kg/m²s,

μ = fluid viscosity at the bulk fluid temperature, Ns/m²,

μ = fluid viscosity at the wall,

C_p = fluid specific heat, heat capacity, J/kg °C

3.3 CALCULATIONS

On the basis of the given problem we will be calculating the values of the basic dimensions of the shell and tube heat exchanger .Only the thermal design will be considered. The Kern's method will be employed for the calculation. As this process is an iterative process, I'm only representing the final iteration of the calculation. As coolant is corrosive, so it is assigned to the tube-side.

Counter Flow Heat Exchanger

Heat capacity water, Cp_h= 4.182 Kj/kg k

Heat load, Q = m_h. c_{ph} $(t_{h1}-t_{h2})$ [1]

= 0.055036 X 4.182 X (100-81.31)

Q = 4.301kw

Heat capacity methanol, C_{pc} = 2.534 kJ/Kg k

The cold and hot stream heat loads are equal so, cooling methanol flow rate is calculated as follow

Cooling methanol flow,

Q = $m_c.C_{pc}$ $(t_{c2}-t_{c1})$ [1]

$$m_c = \frac{Q}{C_{p_c}\left(t_{c_1}-t_{c_2}\right)}$$

$$= \frac{4.301}{2.534(58.22-25)}$$

m_c = 0.0511 kg/s

The well known "logarithmic mean" temperate different (LMTD or θ_m) is calculated by

$$\Delta T_{LMTD} = \frac{\left(t_{h_1}-t_{c_2}\right)-\left(t_{h_2}-t_{c_1}\right)}{ln\left(\frac{t_{h_1}-t_{c_2}}{t_{h_2}-t_{c_1}}\right)}$$

$$\theta_m = \frac{\theta_2-\theta_1}{\ln\left(\theta_2/\theta_1\right)}$$

$$\theta_m = \frac{(100-58.22)-(81.31-25)}{\ln\left(\frac{100-58.22}{81.31-25}\right)}$$

$\theta_m = 48.68^o C$ [1]

Choosing one shell pass and one tube pass for the designing purpose.

Assuming,

U= 140W/m°c

Provisional Area,

Q = U.A.Qm

A= Q/U.Qm

$$A=\frac{4.301\times10^{3}}{140\times48.68}$$

A = 0.6310m^2

Choosing, outer diameter (d_o) 20mm, inner diameter (d_i) 16mm, length of the tube to be 610mm and Aluminium as the material of the tube. The selection of the tube is based on the most easily available size of tube whereas selection of the tube length is arbitrary.

L = 610mm

Surface Area of one tube A $= \pi DL$

= 3.14 X 16 X 10^{-3} X 0.61

= 0.030 m^2

Numbers of tube N_t =

$$\frac{\text{Total outside surface of tubes (Provisional area)}}{\text{Outside surface area of one tube}}$$

$$= \frac{0.6310}{0.030}$$

= 21.033

N_t = 21 Nos.

Tube Pattern Applications

- The triangular and rotated square patterns give higher heat transfer rates, but at the expense of a higher pressure drop than, the square pattern.
- A Square or rotated square arrangement is used for heavy fouling of fluids, where it is necessary to mechanically clean the outside of the tubes.
- As the shell side fluid is relatively clean use 1.25 square pitch.

An estimate of the bundle diameter Db can be obtained from equation below which is an empirical equation based on standard tube layouts. The constants for use in this equation for triangular and square patterns.

$$D_b = d_0\left(\frac{N_t}{K_1}\right)^{1/n_1} \quad [1]$$

Where,

K_1 & n_1 = Constant

K_1 = 0.215

n_1 = 2.207

Bundle diameter

$$D_b = 20\left(\frac{21}{0.215}\right)^{\frac{1}{2.207}}$$

D_b = 145mm

Bundle clearance = 25mm

Use a spilt ring floating head type

From figure 3.3

Bundle diametrical clearance = 25mm

Shell diameter (D_s)

Ds= Db+ clearance

Ds= 145+25

Ds = 170mm

Tube side coefficient

Now we have to calculate the heat transfer coefficient i.e. h_i for the tube side.

Mean water temperature = $\frac{100+81.31}{2}$

= 90.65^{o}C

Tube cross sectional area $a = \frac{\pi}{4}d^2$

$$= \frac{\pi}{4} \times 16^2 = 201mm2$$

Total flow area = 21 X (201 X 10^{-6})

= 4.221 X 10^{-3} m^2

Water mass velocity (G_t) = $\frac{0.055036}{4.221 \times 10^{-3}}$

G_t = 13.03 kg/sm^2

Density of water,(ρ water) = 998.2 kg/m³

Water linear velocity,

$$u_t = \frac{G_t}{\rho_{water}}$$

$$u_t = \frac{13.03}{998.2}$$

u_t = 0.013 m/s

Coefficient for water can also be calculated by using below equation

$$hi = \frac{4200(1.35 + 0.02\,t)\,u_t^{0.8}}{d_i^{0.2}}$$

The above equation has been adapted from data given by eagle and ferquson (1930)

Where,

hi= inside coefficient for water, W/m^{2o}C

T_{avg} = t = water mean temperature ^{o}C

$$hi = \frac{4200(1.35 + 0.02 \times 90.65) \times (0.013)^{0.8}}{(16)^{0.2}}$$

hi = 236.41 W/m^{2o}C

The equation can also be calculated using equation below; this is done to illustrate use of this method.

$$\frac{h_i d_i}{k_f} = j_h \operatorname{Re} \operatorname{Pr}^{0.33} \left(\frac{\mu}{\mu w} \right)^{0.14} \quad [1]$$

Where

hi = inside coeff. (W/m^{2o}C)

j_h = heat transfer factor (dimensionless)

Re = Reynolds number (dimensionless)

Pr = Prandtl number (dimensionless)

μ = Viscosity of water (N s/m^2)

μw = Viscosity of water at wall temperature (Ns/m^2)

Viscosity of water (μ) = 0.8 m N s/m^2

Thermal conductivity (k_f) = 0.59 W/m°C

$$R_e = \frac{\rho \mu d_i}{\mu} = \frac{998.2 X 0.013 X 16 X 10^{-3}}{0.8 \times 10^{-3}}$$

Re = 260

$$P_r = \frac{C_p \mu}{K_f} = \frac{4.182 \times 0.8 \times 10^{-3}}{0.59}$$

Pr =0.0056

Neglect $\mu / \mu w \approx 1$

Put this value: -

$$h_i = \frac{k_f}{d_i} j_h \operatorname{Re}(\operatorname{Pr})^{0.33} \left(\frac{\mu}{\mu w} \right)^{0.14}$$

Coefficient j_h = 3.9 X 10^{-3}

h_i=

$$\frac{0.59}{16\times10^{-3}}\times3.9\times10^{-3}\times260\times(0.0056)^{0.33}\times1)^{0.14}$$

h_i = 6.755 W/m^{2o}C

Shell side coefficient

Now the shell side heat transfer coefficient is calculated i.e. h_s

A close baffle spacing will give higher heat transfer coefficients but at the expense of higher pressure drop.

l_b = Choosing baffle spacing = Ds/5

= 170/5 =34mm

Tube pitch, Pt = 1.25 do

= 1.25 X 20

Pt = 25mm

Area of cross flow,

$$A_s=\frac{(P_t-d_o)D_s l_b}{P_t}$$

$$A_s=\frac{(25-20)\times170\times34\times10^{-6}}{25}$$

$$A_s=1.156\times10^{-3}m^2$$

Mass velocity,

$$G_s=\frac{W_s}{A_s}$$

$$G_s=\frac{0.023675}{1.156\times10^{-3}}kg/\sec \mathrm{m}^2$$

$$G_s=20.48$$

Gs= 20.50 kg/m^2

Where

Ws= methanol flow rate on the shell side kg/sec

Shell equivalent Diameter (hydraulic diameter),

$$d_e=\frac{1.27}{d_o}(p_t^2-0.785d_0^2)$$

$$d_e=\frac{1.27}{20}(25^2-0.785\times20^2)$$

$$\mathrm{d}_e=19.7m$$

Mean shell side temperature,

$$=\frac{58.22+25}{2}$$

$$=41.61$$

$$=42^o c$$

Methanol density(ρ)= 785 kg/m^3

Viscosity, μ= 0.34 mNs/m^2

Heat Capacity , Cpc= 2.534 KJ/Kgk

Thermal Conductivity , K_f = 0.19 W/m ºC

$$\mathrm{Re}=\frac{Gsde}{\mu}$$

$$=\frac{20.50\times19.7\times10^{-3}}{0.34\times10^{-3}}$$

Re= 1187.79

Re= 1188

$$\mathrm{Pr}=\frac{C_{p_c}\times\mu}{K_f}=\frac{2.534\times10^3\times0.34\times10^{-3}}{0.19}$$

Pr= 4.5

Crossing 15% baffle cut

Heat transfer factor

i_h = 4.6 X 10^{-2}

Without the viscosity correction term,

$$Nu = \frac{h_s d_e}{k_f} = j_h \operatorname{Re} \Pr^{1/3} \left(\frac{\mu}{\mu w} \right)^{0.14}$$

Neglect , $\frac{\mu}{\mu w}$ =1

$$h_s = \frac{0.19}{19.7 \times 10^{-3}} \times 4.6 \times 10^{-2} \times 1188 \times (4.5)^{1/3}$$

h_s = 870.16W/m$^{2\,0}$C

Estimate wall temperature,

Mean temperature difference,

Across all resistance = 90.65 -42

= 48.65^{o}C

Across methanol film = $\frac{U}{h_s} \Delta T$

$$= \frac{140}{870.16} \times 48.6$$

= 7.8^{o}C = 8^{o}C

Mean wall temperature =42-8

= 34^{o}C

μw =0.37m N s/m^2

$\left(\frac{\mu}{\mu w} \right)^{0.14}$ = 0.988

= 0.99

Which shows that the correction for a low viscosity fluid is not significant

Overall Coefficient

Thermal Conductivity of aluminium alloys = 237 W/m ^{o}C

Taking the fouling coefficients from table 3.1 methanol (light Organic) 5000 Wm^{-2o}C^{-1}, brackish water (sea water), take as highest value, 3000 Wm^{-2o}C^{-1}

$$\frac{1}{U_o} = \frac{1}{h_s} + \frac{1}{h_o d} + \frac{d_o \ln \frac{dn}{di}}{2k_w} + \frac{do}{di} \times \frac{1}{h_i} + \frac{do}{di} \times \frac{1}{h_i}$$

$$\frac{1}{U_o} = \frac{1}{870.10} + \frac{1}{5000} + \frac{20 \times 10^{-3} \ln(20/16)}{2 \times 50} + \frac{20}{16} \times \frac{1}{3000} + \frac{20}{16}$$

U$_o$ =140.88 w/m^{2o}c

Above assumed value of 140 W/m ^{o}C

Dimensions of Shell and Tube Heat Exchanger

Tubes:

1. Outer dia. = 20 mm
2. Inner dia. = 16 mm
3. Length of the tubes = 610 mm
4. No. of the tubes = 21 tubes
5. Bundle tubes = 145 mm
6. Bundle dia. Clearance = 25 mm

Shell:

1. Inner dia. = 170 mm
2. Baffle spacing = 34 mm
3. Baffle thickness = 2 mm
4. No. of baffle plates = 25

4. RESULTS & DISCUSSIONS

In beginning of the project we calculated the design values for our heat exchanger. Those calculated values were used in the CFD simulation for the analysis of our heat exchanger, now to study our design through

Ansys we studied the temperature pressure and velocity profiles for our heat exchanger.

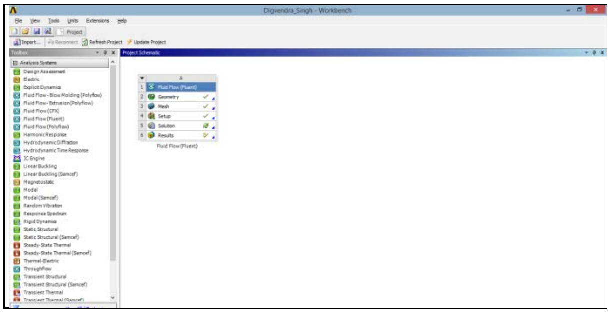


Figure 4.1 Project Schematic

4.1 Design Modeler

This is Ansys design modular environment. Here the physical geometry of the heat exchanger can be drawn or imported from another CAD software like CREO, CATIA, SOLIDWORKS, UNIGRAPHICS, etc. The heat exchanger modelled here contains 21 tubes each 610 mm long. We have also modelled internal and external fluids. Since this is a symmetrical type model we have shown symmetrically cut section model and this will reduce computation time without affecting the results.

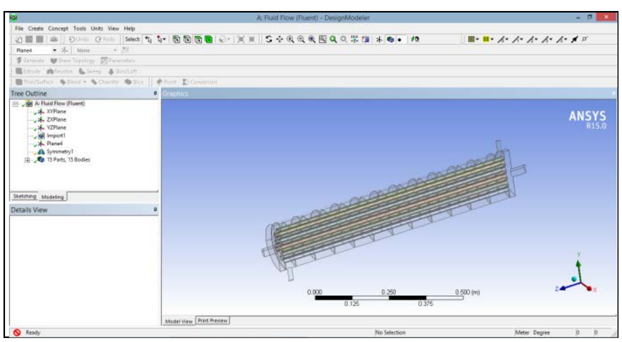
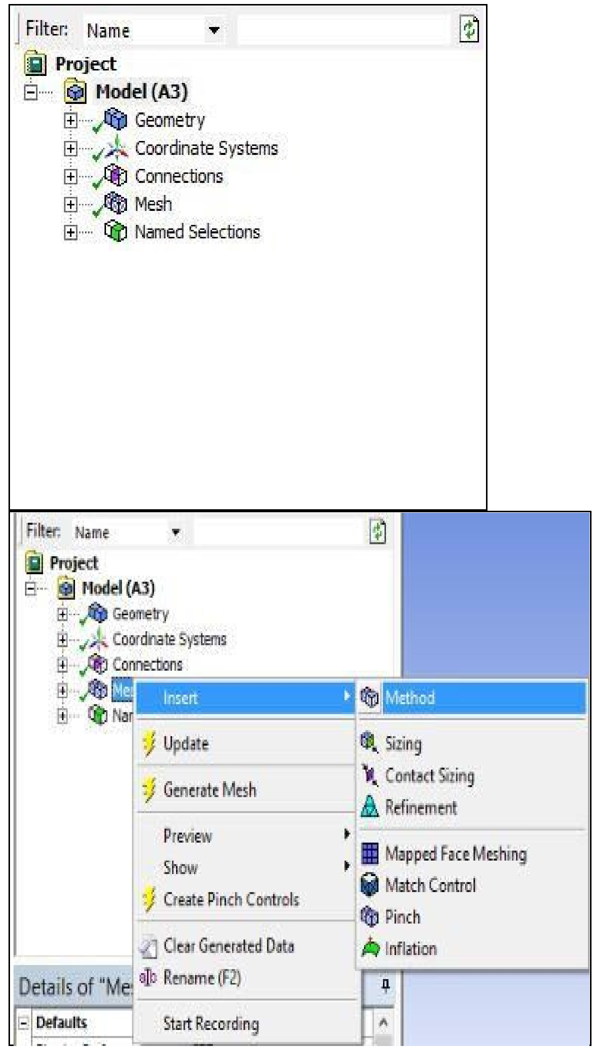
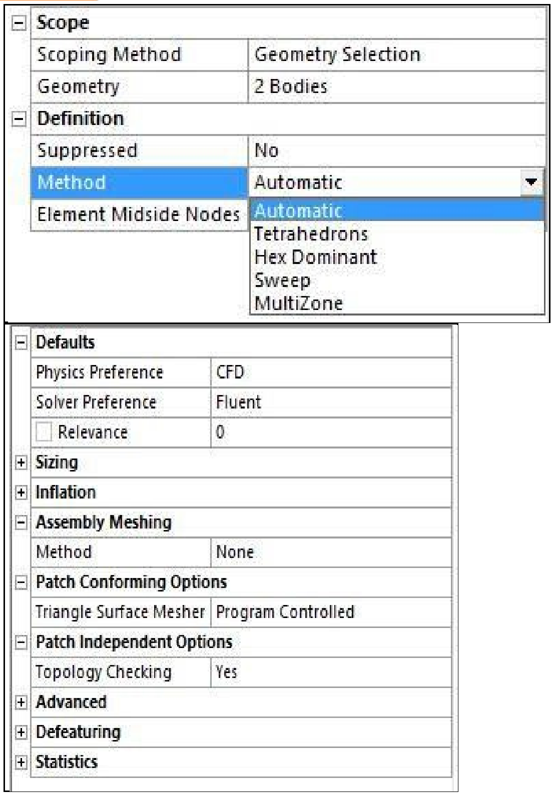


Figure 4.2 Design Modeler

4.2 Meshing



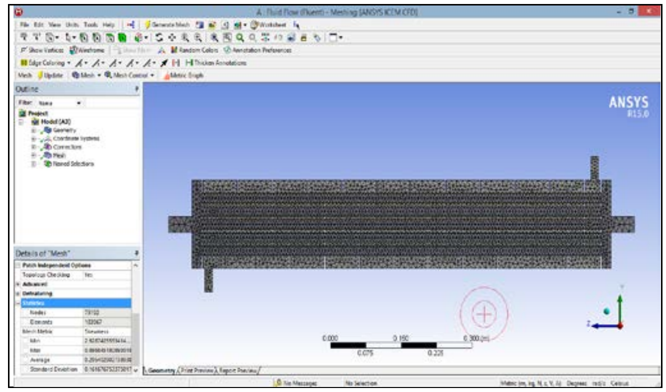
(a)



(c)

Figure 4.3 Meshing Process

Figure 4.3 (a) shows us the meshing tool window. From figure b and c we can see that we can add the method we want to use in the meshing process which in my case is the sweep method. Figure d shows us the bunch of other options which are also very useful but they need very good configuration computers as they make a very fine quality of mesh and it results in higher no. of nodes and elements resulting in a better and complex solution. Figure 4.4 shows us the generated mesh designed by using this process.



(d)

Figure 4.4 the mesh generated for the geometry of the shell and tube Heat Exchanger

This is Ansys meshing environment. Here the CAD model of heat exchanger is divided into several numbers of nodes and elements. Meshing is required to get the solution at different locations of the heat exchanger. The solver will calculate for each node (using iterative method) which is created by meshing. This mesh was generated by Ansys meshing. It will be same for all cases and only boundary conditions will vary. From the Figure 5.4 it can be observed that the mesh contains 75192 nodes and 183967 elements. The finer is the mesh, the smaller is size of elements. This will result in more computation time also. This mesh is an unstructured one and was tested for grid independency. We have found that refining of mesh to more smaller divisions are not required for this study as further refining will not improve solution to more than 5%. We have also found that meshing the component to coarse mesh would cost accuracy of the solution. So current mesh can be acceptable. The mesh metric skewness is also less than 0.9. It is calculated by the Ansys software, if metric skewness is more than 0.9 than, meshing are not possible.

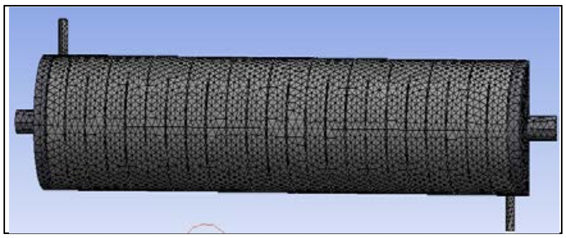


Figure 4.5 the mesh generated for the geometry of the shell and tube Heat Exchanger

4.3 Solution

The setup in fluent is used to generate the solution using the boundary conditions. Here is a pictorial representation of the process used in setting up the fluent setup.

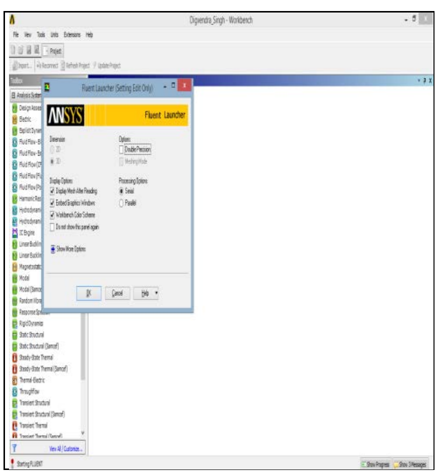


Figure 4.6 Fluent Launcher

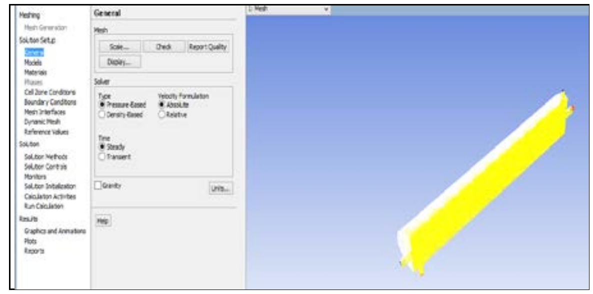


Figure 4.7 Fluent setup (General)

This is the Ansys Fluent solver environment. Here solution method and boundary conditions are specified. Before beginning with solution steps, the general settings are to be addressed. Press check button in mesher section to check the mesh produced by ansys mesher. The quality of incoming mesh is obtained by pressing report quality button in mesher section.
We also need to set the units of operating variables like length, temperature, velocity, pressure, density, heat flux, etc.

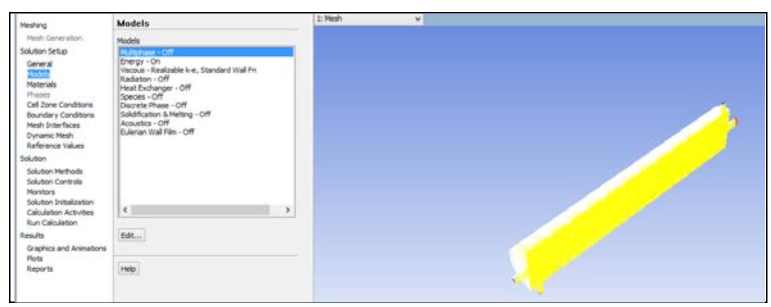


Figure 4.8 Fluent setup (Models)

Figure 4.6, 4.7 and 4.8 shows us the starting windows of the fluent. As we can see there are all the options available for the CFD simulation in those windows. We start from the energy equations selection from the above window as it is applicable in our heat exchanger model. Then

International Journal of Scientific Engineering and Applied Science (IJSEAS) – Volume-2, Issue-3,March 2016
ISSN: 2395-3470
www.ijseas.com

next comes the turbulence model section. As I have gone through different papers which suggested the k-epsilon model to be the best and effective method for a heat exchanger evaluation. So I have also selected this model due to its greater accuracy in heat exchanger cases from model settings, turn on the energy equation. Also enable the viscous settings to k-epsilon realizable settings and enhance wall functions.

After the equations selection comes the material selection based on the problem which in our case is methanol water and aluminium.

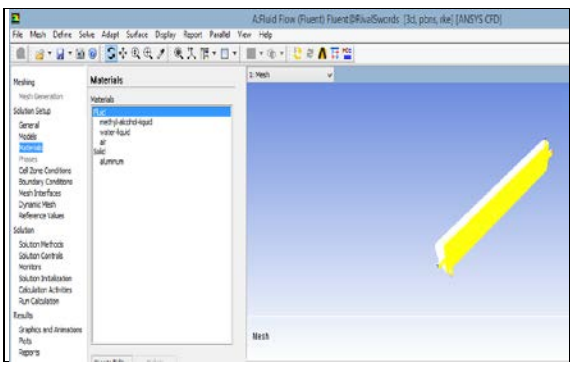


Figure 4.9 Material Selection

Here materials are assigned to the parts modelled in cad software. Material assigned to cold fluid is methanol. It is shell side fluid. While material assigned for hot fluid is water. It is tube side fluid. Both of these materials are present in Ansys Fluent's material library.

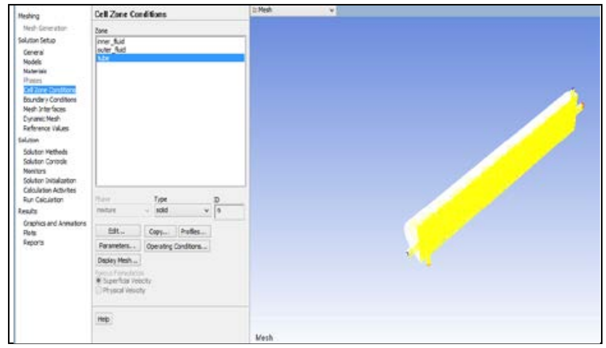


Figure 4.10 Material Selection (tubes)

Now we need to define cell zone conditions. Here the solid/ fluid behaviour of material is defined.

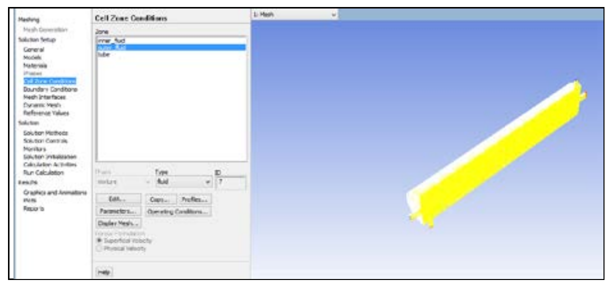


Figure 5.11 Material Selection (outer fluid)

Boundary Conditions:

Boundary conditions are used according to the need of the model. The inlet temperatures and velocities are given to the setup.

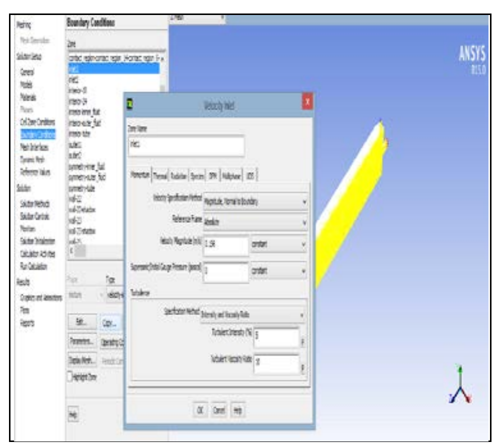


Figure 4.12 Boundary condition of the fluids

Now boundary conditions are defined. This analysis will be carried out for counter flow conditions. We have used 7 different boundary conditions for our heat exchanger which are tabulated below:

Table 4.1 Boundary Conditions

Case . No.	Inlet Velocity of hot fluid(water)(in m/s)	Inlet Temp of hot fluid (water) (in Kelvin's)	Inlet Velocity of cold fluid (methanol)(in m/s)	Inlet Temp of cold fluid (methanol)(in Kelvin's)
1	0.156	368	0.3	298
2	0.1	373	0.3	298
3	0.156	373	0.3	293
4	0.156	373	0.3	298
5	0.2	373	0.3	298
6	0.3	373	0.3	298
7	0.3	373	0.5	298

Ansys Fluent Solution will be used to simulate solutions for all these conditions.

Now solution methods are set up. Here second order upwind scheme is chosen in spatial discretization section for momentum, pressure, turbulent kinetic energy, energy and turbulent dissipation rate.

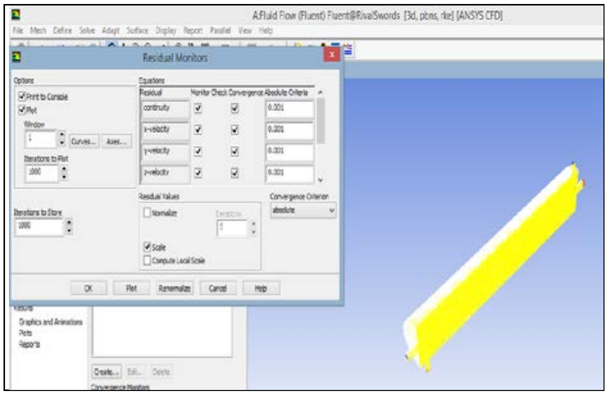


Figure 4.13 Inlet temperature and velocity condition of the fluids

Convergence of solution is defined at Residual monitor's settings. The degree of solution accuracy will be defined here.

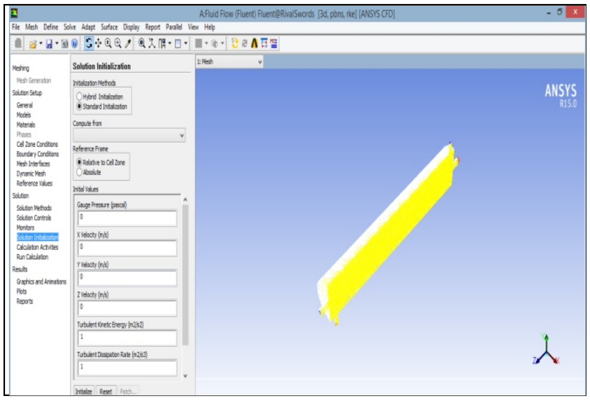


Figure 4.14 Solution is initialized

Then the solution is initialized.

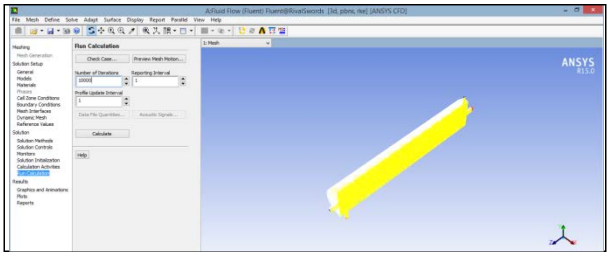


Figure 4.15 Run calculation

Now the solution is calculated. Numerical iterations are performed in this section till acceptable solution is reached.

4.4 Contour Plots

Solution for case 1

Conditions are:

- Inlet Velocity of hot fluid (water) = 0.156 m/s
- Inlet temperature of hot fluid (water) = 368 K
- Inlet Velocity of cold fluid (methanol) = 0.3 m/s

- Inlet temperature of cold fluid (methanol) = 298 k

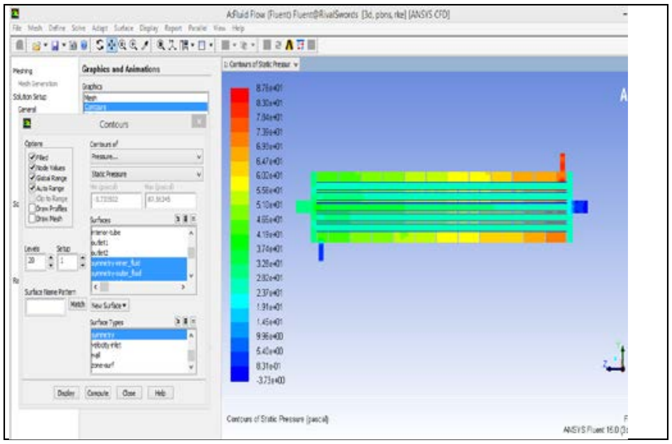


Figure 4.16.1 Pressure contour plot at symmetrical plane

These are the contours of pressure for case 1. Pressure on any location can be obtained in Pascal by matching the colour on that location with colour of colour scale.

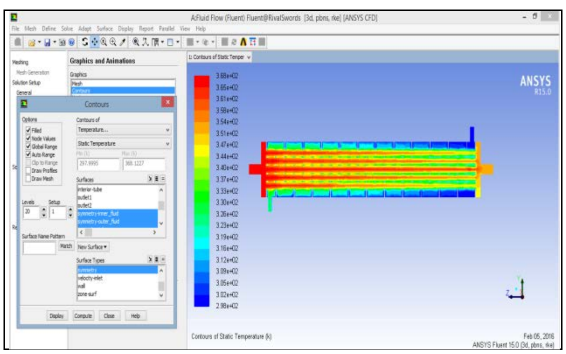


Figure 4.16.2: Temperature contour plot at symmetrical plane

These are the contours of temperature for case 1. Temperature of fluid/tubes on any location can be obtained in Kelvin's by matching the colour on that location with colour of colour scale.

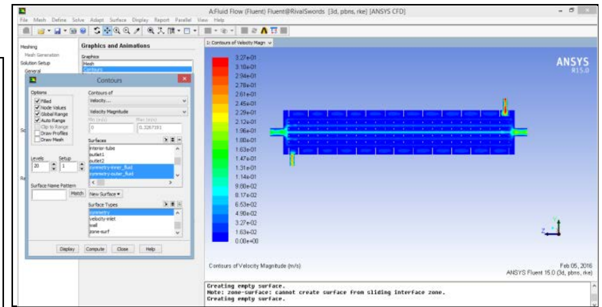


Figure 4.16.3: Velocity contour plot at symmetrical plane

These are the contours of velocity for case 1. Velocity of any fluid on any location can be obtained in m/s by matching the colour on that location with colour of colour scale.

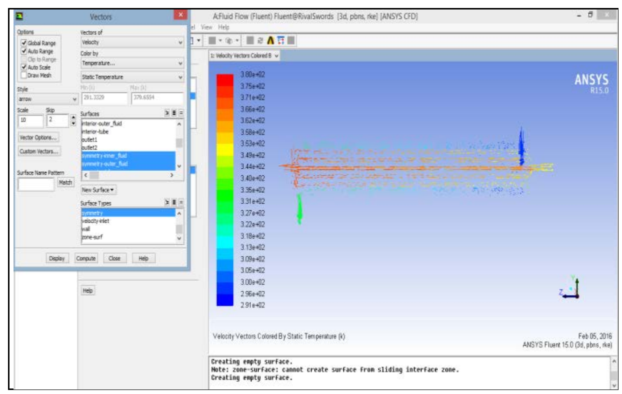


Figure 4.16.4 Velocity vectors contour plot at symmetrical plane

Table 4.2 CFD Results for temperature and the overall effectiveness

Simulation Case. No.	Inlet Velocity of hot fluid (water) (in m/s)	Inlet Temp of hot fluid (water)(in Kelvin's) $T_{h,max}$	Outlet Temp of hot fluid (water)(in Kelvin's) $T_{h,min}$	Inlet Velocity of cold fluid(methanol) (in m/s)	Inlet Temp of cold fluid (methanol)(in Kelvin's) $T_{c,min}$	Outlet Temp of cold fluid (methanol)(in Kelvin's)	Effectiveness (%)
1	0.156	368	350.56552	0.3	298	328.99466	95.583
2	0.1	373	346.65433	0.3	298	330.34497	86.403
3	0.156	373	353.70276	0.3	293	330.74741	92.546
4	0.156	373	354.31256	0.3	298	331.22256	95.595
5	0.2	373	359.0162	0.3	298	332.51306	91.707
6	0.3	373	364.45981	0.3	298	334.12851	84.0116
7	0.3	373	359.61053	0.5	298	328.44592	79.028

These are the velocity vectors for case 1. These vectors are coloured by temperature. Temperature of fluid molecules on different locations is obtained by matching the colour on that location with colour of colour scale.

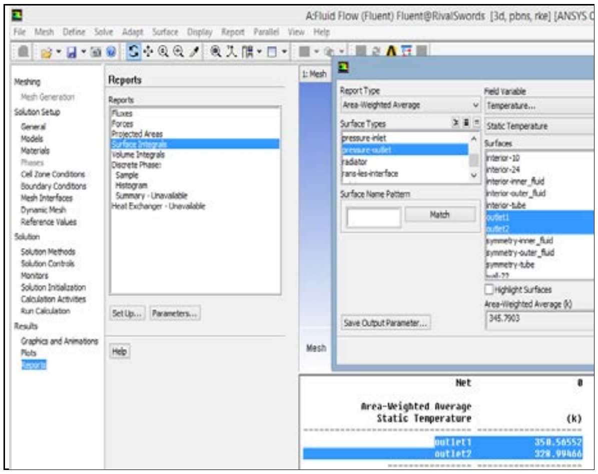


Figure 4.16.5 Average temperatures calculated

These are the average temperatures calculated for outlet zones of hot and cold fluids. Outlet 1 represents temperature of hot fluid when it leaves the heat exchanger. Outlet 2 represents temperature of cold fluid when it leaves the heat exchanger.

From above figure we can clearly see significant changes like temperature drop of hot fluid and temperature rise of cold fluid at exit zones. In a similar way exit temperatures will be calculated for all 7 cases.

Results:

The given heat exchanger analysis was carried out under counter flow conditions. The simulated results are tabulated below for 7 different conditions which were mentioned in previous table also.

In current manufacturing process in the steel industries water is being used both as shell and tube side in heat exchanger. The shell side fluid is water exit from the heat exchanger and goes to cooling tower for cooling of fluid but water is not sufficient cooled by the cooling tower for again using in heat exchanger.

The new designing dimension of a shell and tube heat exchanger having 21 tubes, 170mm shell diameter and 610 long in this design does not include the type of the header to be used.

After modifying the heat exchanger the effectiveness of heat exchanger is increased by changing the shell side fluid (from water to methanol). Methanol provide sufficient cooling to as compared water in heat exchanger and **increase the effectiveness of heat exchanger up to 95.5%.**

5. Conclusion

Kern's method is the most used and effective method for the designing procedure. In the research analysis shell and tube heat exchanger is designed and simulated by using Computational Fluid Dynamic (CFD).The header selection for the heat exchanger has also been based on the Computational Fluid Dynamic (CFD) simulation. I can see that the uniform flow in tubes can be achieved using a suitable header. The nozzle placement normal to the plane of tubes and also eccentric to the head side of the headers has been the most effective. The simplified geometry of the shell and tube heat exchanger is used. The assumption of plane symmetry works for most of the length of heat exchanger except the outlet and inlet regions where the rapid mixing and change in flow direction takes place. Thus improvement is expected if complete geometry is modeled. Furthermore, the enhanced wall functions are not used in this project due to convergence issues, but they can be very useful with k−epsilon models. The heat transfer is found to be poor because the most of the shell side fluid by-passes the tube bundle without interaction. Thus the design can be modified in order to achieve the better heat transfer in two ways. Either, the shell diameter is reduced to keep the outer fluid mass flux lower or tube spacing can be increased to enhance the inner fluid mass flux. Just doing this might not be enough, because it is seen that the shell side fluid after 550 mm doesn't transfer heat efficiently.

Then, change the length of tube up to 610 mm from 550 mm.The new designing dimension of a shell and tube heat exchanger having 21 tubes, 170mm shell diameter and 610 long in this design does not include the type of the header to be used. After modifying the heat exchanger the effectiveness of heat exchanger is increased by changing the shell side fluid (from water to methanol). Methanol provide sufficient cooling to as compared water in heat exchanger and increase the effectiveness of heat exchanger up to 95.5%.

REFERENCES

[1] P.K Nag, Heat and Mass Transfer,McGraw Hill education (India) Private Limited,New Delhi.

[2] Frank P. Incropera, David P. Dewitt, Theodore L. Bergman, Adrienne S. Lavine "Fundamentals Of Heat And Mass Transfer", John Wiley & Sons SIXTH EDITION, 111 River Street, Hoboken, NJ 07030-5774 2007.

[3] http://www.wcr-regasketing.com, **"Heat exchanger applications"** http://www.wcr-regasketing.com/heat-exchanger-applications.htm, 2010.

[4] P M V Subbarao **"Figure: Arrangement of tubes, Tube Layout and Flow Scales" mel709 tutorial**, Professor m*echanical Engineering Department* ***I I T Delhi.***

[5] Majid Hayati **"Kern technique of Shell-And-Tube Heat Exchanger"**.University of Kashan, Kashan, I.R. IRAN 2014.