Heat Transfer Enhancement in Heat Exchangers using Customized Turbulators

1. Swapnil D. Shelar
Engineering Student,
Dept. Mechanical Engineering.
Sinhgad Academy of Engineering
Pune, India

2. Shashank P. Dashpute
Engineering Student,
Dept. Mechanical Engineering.
Sinhgad Academy of Engineering
Pune, India

Abstract— **Heat transfer augmentation techniques are used to increase rate of heat transfer without affecting much the overall performance of the system. Heat transfer augmentation techniques are commonly used in areas such as heating and cooling in evaporation, air–conditioning equipment, thermal power plants, space vehicle, automobile etc. Heat transfer enhancement by creating turbulence in the physical behaviour of fluid flow inside the heat exchanger tube has become a really intresting area for the researches in order to reduce the size and cost of the involving equipment especially in heat exchangers. This Project Report represents the effect of the and Conical Circular Ring (CCR) on the heat transfer and fluid friction characteristics in a heat exchanger tube. The Numerical Analysis were conducted in CFD with and CCRs of various geometries, including three different diameter ratios (DR=d/D=0.6, 0.7 and 0.8). Hot air is passing in tube under constant heat flux condition in the present work. The Reynolds number was varied from 8000 to 18,000.**

Keywords— *Heat Transfer; Reynolds Number; Thermal Enhancement factor; Customized Turbulators.*

I. INTRODUCTION

Heat exchangers are used in different processes ranging from conversion, utilization & recovery of thermal energy in various industrial, commercial & domestic applications. Some common examples include steam generation & condensation in power & cogeneration plants; sensible heating & cooling in thermal processing of chemical, pharmaceutical & agricultural products; fluid heating in manufacturing & waste heat recovery etc. Increase in Heat exchanger's performance can lead to more economical design of heat exchanger which can help to make energy, material & cost savings related to a heat exchange process. The need to increase the thermal performance of heat exchangers, thereby effecting energy, material & cost savings have led to development & use of many techniques termed as Heat transfer Augmentation. These techniques are also referred as Heat transfer Enhancement or Intensification. Augmentation techniques increase convective heat transfer by reducing the thermal resistance in a heat exchanger. Use of Heat transfer enhancement techniques lead to increase in heat transfer coefficient but at the cost of increase in pressure drop.

II. CLASSIFICATION OF ENHANCEMENT TECHNIQUES

1. Passive Techniques:-

    These techniques generally use surface or geometrical modifications to the flow channel by incorporating inserts or additional devices. They promote higher heat transfer coefficients by disturbing or altering the existing flow behaviour (except for extended surfaces) which also leads to increase in the pressure drop. In case of extended surfaces, effective heat transfer area on the side of the extended surface is increased.
    1.1. Extended Surfaces
    1.2. Displaced Enhancement Services
    1.3. Swirl Flow Devices
    1.4. Coiled Tubes

2. Active Techniques:-

    These techniques are more complex from the use and design point of view as the method requires some external power input to cause the desired flow modification and improvement in the rate of heat transfer.
    2.1. Mechanical Aids
    2.2. Surface Vibration Techniques
    2.3. Fluid vibration
    2.4. Electrostatic Fields

3. Compound Techniques:-

    A compound method is a hybrid method in which both active and passive methods are used in combination. Two or more techniques may be utilized simultaneously to produce an enhancement.
    3.1. Rough tube wall with twisted tape
    3.2. Rough Cylinder with acoustic vibrations
    3.3. Internally Finned tube with twisted tape insert
    3.4. Finned tube with fluidized beds

III. EXERIMENTAL SETUP AND CALCULATION:-

The details of the experimental facility are demonstrated in fig 1 The apparatus consists of 0.5HP suction type blower, an orifice flow meter, a heat exchanger tube (test section), a U-tube manometer, a micro-manometer, a set of temperature measurement devices (thermocouple), a variance transformer, a watt meter, nichrome wire for heating the test section with asbestos coating for insulation purpose. The tube is made from mild steel because of its high thermal conductivity. For heating the test section and to provide a uniform heat flux boundary condition, a continually wound flexible electrical wire is provided. It is a nichrome wire, provided with asbestos coating of thickness 5 cm as an insulation to minimize the convective heat loss to the surroundings.

1. COMPONENTS:-
    1.1. Bell Mouth Inlet
    1.2. Calming Section
    1.3. Test Section
    1.4. Thermocouples
    1.5. Wattmeter
    1.6. U tube Manometer and Micro Manometer

2. WORKING:-
    The air is sucked in a calming section with a bell mouth inlet by a blower with capacity 0.5 HP. After the air is fully developed in the calming section it enters the test section. The test section is provided with the circumferential heater along its length. The thermocouples are provided on the test section to measure the temperature of the heated air, due to the circumferential heater, only the layer of air in contact with the inner diameter of the pipe gets heated up and the air at the center is slightly heated up. To avoid such condition at outlet to get correct reading, baffles are inserted at the outlet of the pipe to maintain the uniform temperature of air by thoroughly mixing it after outlet so as to get the correct output temperature reading. The pressure drop across the tube is calculated with the help of the U-tube manometer. For the Reynolds number ranging from 8000 to 18000, the heat transfer and the friction factor is calculated for the plain tube as well was for the inserts with 0.7dr, 0.8dr and 0.6dr blockage area.
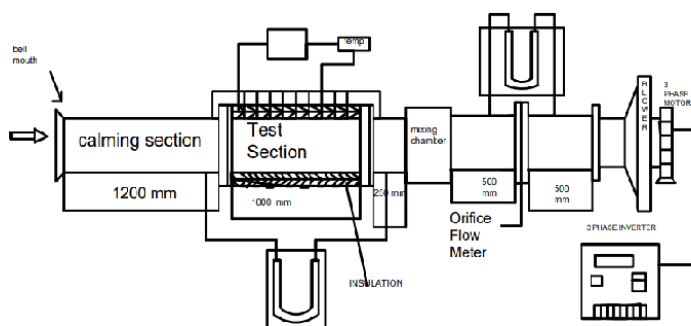

Fig 1. Schematic Diagram

3. PRE and POST EXPERIMENTAL DATA REDUCTION:-

These are the calculations need to perform before conduction the experiment

3.1. Reynolds Number (Re):
    Reynolds number is a parameter which indicates the flow regime of the working fluid and defines as

    Reynolds number $(Re) = (V*d)/\upsilon$
    Where, υ= Kinetic viscosity of working fluid (m²/s)
    d= Diameter of pipe (m)
    V= Flow velocity (m/s)

3.2. Wall surface temperature(Tw):-
    It is the average inner wall surface temperature measured by ten thermocouples. It is calculated by taking average of ten thermocouple temperature readings $T2$ to $T11$
    $T_w = (T_2 + \ldots + T_{11})/10$

3.3. Bulk mean temperature(T_{bm}):-
    It is also called average bulk temperature. It is a convenient reference point for evaluating Properties related to convective heat transfer.
    $Tbm = (T_{in} + T_{out})/2$
    Where, A = Heat transfer area = Inner surface area of pipe (mm2)
    Tw = Average inner surface temperature of test tube (0C)
    Tbm = Bulk temperature of air (0C)
    Tin = Inlet air temperature = $T1$ (0C)
    $Tout$ = Outlet inlet temperature = $T13$ (0C)
    Cp = Specific heat of air at Tbm (KJ/kg-k)
    $\dot{m}$ = Mass flow rate of air (kg/s)

3.4. Nusselt Number:-
    The Nusselt number is a measure of the convective heat transfer occurring at the surface and is defined as
    Nusselt number (Nu)= $(h*d)/k$
    Where h = Convective heat transfer coefficient (w/m2k)
    d = Diameter of the tube (m)
    k = Thermal conductivity (w/m-k)

3.5. Friction Factor:-
    Friction Factor, a dimensionless parameter f is written as,
    Friction factor $(f) = \Delta P/[(L/D)*(1/2*\rho*V^2)]$
    Where, ΔP = Pressure across test section (pa)
    $\Delta P = \rho diff * g * \Delta h$
    $\Delta P = 47 * g * \Delta h$ for micro-manometer
    $\Delta P = 997 * g * \Delta h$ for U-tube manometers

3.6. Thermal performance factor(η):-
    It is defined as the ratio of the heat transfer enhancement ratio to the friction factor ratio,
    TPF (η) = $[(Nu_t + Nu_p)/(f_t/f_p)^{1/3}]$

Where Nu_t = Nusselt number for pipe fitted with turbulator rings

f_t = Friction factor for pipe fitted with turbulator rings

Nu_p = Nusselt number for plain or smooth pipe

f_p = Friction factor for plain or smooth pipe

IV. METHOD OF DETERMINING SOLUTIONS TO FLOW PROBLEMS:-

Computational Fluid Dynamics:-

Computational Fluid Dynamics (CFD) is the use of computer-based simulation to analyze systems involving fluid flow, heat transfer and associated phenomena such as chemical reaction.

A numerical model is first constructed using a set of mathematical equations that describe the flow. These equations are then solved using a computer program in order to obtain the flow variables throughout the flow domain.

CFD Procedure Followed:-

- Pre-processor:-

  Pre-processing is the input of a flow problem to a CFD program by means of an operator friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The steps involved in this are:

  1. Definition of geometry.
  2. Grid generation.
  3. Selection of the phenomena or system to be modeled.
  4. Definition of fluid properties.
  5. Boundary conditions specification.

- Solver:-
  1. Approximation of the unknown flow variables by means of simple functions.
  2. Discretization by substitution of the approximations into the governing flow equations.
  3. Solution of algebraic equations.

- Post-processor:-

  All the leading CFD packages are equipped with versatile data visualization tools. These include:-
  1. Domain geometry and grid display
  2. Vector plots.
  3. Line and shaded contour plots
  4. Color postscript output.

- The mathematical modeling of a flow problem is achieved basically through three steps:
  1. Developing the governing equations describing the flow.
  2. Discretization of the governing equations.
  3. Solve the resulting numerical equations.

- Governing equations:-
  1. *Unsteady state 3-D equation of continuity*:-
  $(d\rho/dt) + \Box \cdot (\rho.v) = 0$

Where ρ is the fluid density, t is time, and v is velocity.

2. *Momentum equation*:-

$\rho*(DV/Dt) = \nabla.\tau ij - \nabla p + \rho F$

Where P is pressure

3. *Energy equation*:-

$d(\rho i) + div(\rho iu) = -Pdivu = div(KgradT) + Si + \phi$

V. NUMERICAL INVESTIGATION FOR PLAIN TUBE AND WITH INSERT:-

The numerical analysis is of Air flowing through a horizontal circular pipe at constant wall temperature, for estimation of the heat transfer and friction factor characteristics. The results obtained from numerical investigations are validated by comparing with the available correlations.

1. Plain Pipe:-

   Consider the problem of air flowing through a circular pipe with length (L) and inside diameter of 1800 mm and 48.5 mm respectively.

2. Pipe with Ring inserts:-

   The inlet and outlet Diameter of the pipe with the circular rings inserts and with diameter ratio of 0.6, 0.7&0.8dr is given in the following table:-

TABLE 1. CCR Geometry Specification Table

Quarter CCR geometry	Di(mm)	Do(mm)
0.6 DR	27.9	46.5
0.7 DR	32.55	46.5
0.8 DR	37.2	46.5

The Rings are inserted in the Test section of the pipe. The Length of the test section is 1000mm so there are 20 Rings in total with the first ring placed at 39.25 mm from the calming section. Conical rings are 3mm wide and are placed 48.5 mm apart.
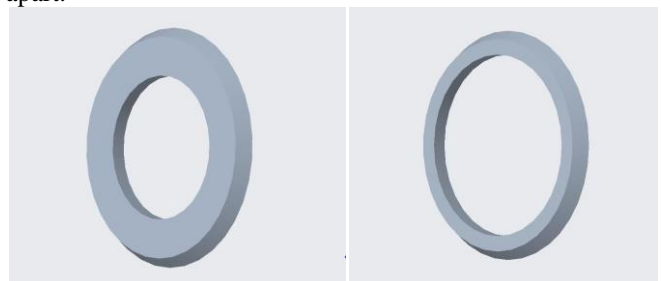

Fig 2. Conical Circular Ring CAD Model

3. FLUENT Simulation:-

   3.1. Geometry Creation and Mesh generation:-

   A circular pipe of diameter 48.5 mm and length 1950 mm was used as the geometry. A 3-D geometry is created by using ANSYS workbench. Proximity and Curvature sizing is used for meshing the geometry. It is meshed as shown in Figures below. Total number of nodes was used 607409 and total number of element used 1660258.
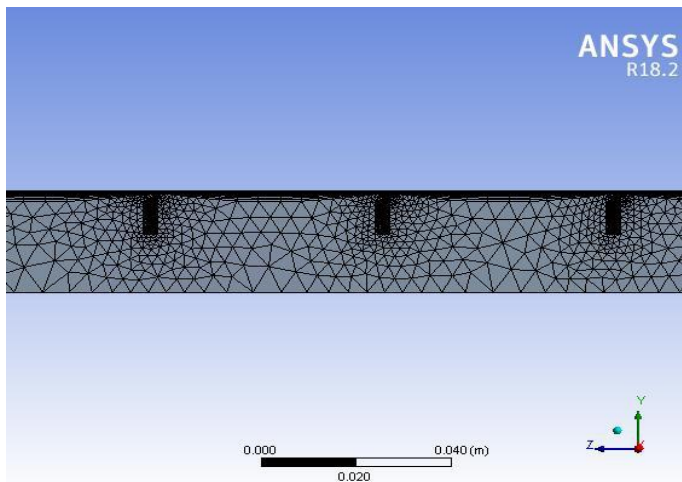
Fig 3. Mesh Body Sectional View

From Grid Independency we check the accuracy of the computational grid for calculating the Nusselt number and friction factor through the heat exchanger Pipe. From the data we get to know that the number of elements varies between 15 lakh & 20 lakh depending on the values selected in sizing function of mesh.

3.2. FLUENT Setup:-

3.2.1.   Modeling Strategy and Solution Method:-
ANSYS Fluent 18.2 is selected for the numerical analysis by using finite volume method with SIMPLE algorithm. The flow is three-dimentional and incompressible, and quasi- steady flow condition. The momentum, energy, turbulent kinetic energy, and dissipation rate are discretized by using second-order upwind scheme. The residuals of continuity, momentum, k and ε equations for the convergence of the numerical computations are set to be less than 10^{-6} and 10^{-9} for the energy equation. Four different turbulence models were employed for the numerical simulations (RNG k-e, standard k-e, Realisable k-e and SST k-w model). The k-e RNG turbulence model with enhanced wall treatment is selected for Numerical Simulation of the pipe with inserts.

3.3. Defining the Material Properties:-
Under materials Tab in FLUENT. The selection of material is done. Material selected is Air. The properties of Air is taken as follows:-

TABLE 2. Properties of hot air

Sr. No	Properties	Value
1	Density	1.225 kg/m3
2	Specific heat	1.0063 kJ/kgK
3	Thermal conductivity	0.025219 W/mK
4	Dynamic viscosity	1.7965x10^-5 Kg/ms

3.4. Boundary Conditions:-
Hot Air was used as the working fluid. The numerical studies were carried out with uniform velocity profile at the inlet of pipeline. The direction of the flow was defined normal to the

boundary. Inlet velocity and temperature were specified. Turbulent intensity, I and the hydraulic diameter, Dh were specified as 5% and 0.0485m of turbulent (k and ε). Outflow boundary condition was used at pressure-out let boundary. The wall of the pipe was assumed to be perfectly smooth with zero roughness height and stationary wall, no slip condition. Initial outlet gauge pressure was taken as zero Pascal. Temperature was taken as 300 K. Constant Heat flux of 1500 $W/m2$ on the outer surface of the horizontal tube is applied. Velocity-Inlet Boundary: inlet was taken for the pipe inlet section and the velocity values for the different Reynolds Number are –

TABLE 3. Velocity for Different Reynolds Number

Reynolds Number	Inlet Velocity (m/s)
Re 8000	2.437
Re 10000	3.046
Re 12000	3.655
Re 14000	4.265
Re 16000	4.874
Re 18000	5.484

3.5. Executing the FLUENT Code:-
Each case must be initialized before the FLUENT code begins iterating towards a converged solution. In this study, the option chosen for initialization was to compute from inlet zones. The number of iterations ranged between 800-12000 depending on the case being run and how long it took to converge. Iterations were performed with the default under- relaxation factors for all parameters.

4.   Data Reduction:-
- Heat added to water was calculated by,
  Q= m cp (To- Ti)
- Heat transfer coefficient was calculated from,
  h = q/(Tw−Tb)
- And heat flux was obtained from,
  q = (Q/As)
  Where, As= π D L
- The bulk temperature was obtained from the average of water inlet and outlet temperatures,
  Tb = (Ti+To)/2
- The volume flow rate Q is given by
  Where, Q = Ac V
  $Ac = \pi D^2/4$
- The Mass Flow rate was obtained from,
  m = Q ρ
- The Reynolds number (Re) and Nusselt number (*Nu*), can be defined as
  Re= ρVD/μ
  Nu=hD/k

  where h is the local heat transfer coefficient, *k* the thermal conductivity of water *u* the inlet velocity, Tw is the wall temperature and Tb the bulk temperature of the fluid flow. The friction factor (f)

and thermal performance factor (η) are defined as follows:-

$f = (2D \, \Delta P)/(L \, \rho V^2)$

$\eta = [(Nu/Nu_s)/(f/fs)^{1/3}]$

Where Nus and fs are the corresponding values obtained from the smooth pipe at the same Re.

The standard correlations include Dittus-Boelter and Blasius for the fully developed turbulent flow in circular tube

Nusselt number correlation: Dittus-Boelter correlation:

Nu =0.023 Re$^{0.8}$Pr$^{0.3}$

Friction factor correlation : Blasius correlation:

f = 0.316 Re$^{0.25}$

VI.   NUMERICAL ANALYSIS:-

1.   Grid Dependency:-

Grid independence test a solution refinement method is used. By using solution-adaptive refinement, cells can be added where they are needed in the mesh, thus enabling the features of the flow field to be better resolved. When adaption is used properly, the resulting mesh is optimal for the flow solution because the solution is used to determine where more cells are needed. In other words, computational resources are not wasted by inclusion of unnecessary cells. Furthermore, the effect of mesh refinement on the solution can be studied without completely regenerating the mesh. For grid independence test a solution adaptive refinement method is used. By using solution adaptive refinement, cells can be added where they are needed in the mesh, thus enabling the features of the flow field to be better resolved. When adaption is used properly, the resulting mesh is optimal for the flow solution because the solution is used to determine where more cells are needed. Meshing of the domain is done using ANSYS software. Computational domain was discretized using tetrahedral elements of different sizes. In order to resolve the flow near the wall accurately prism layers were added.
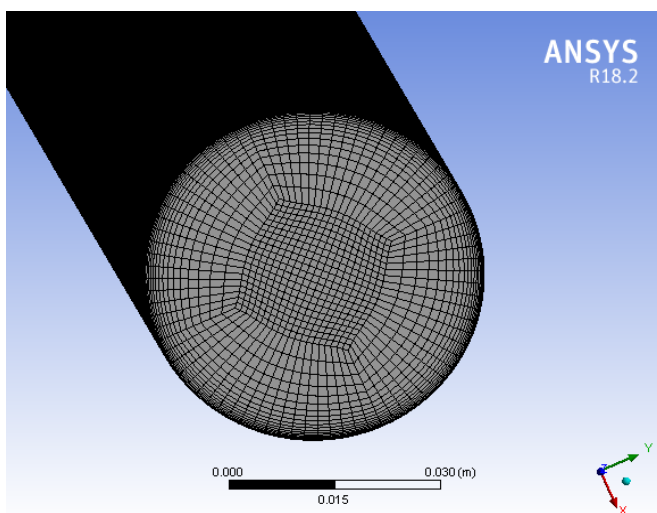

Fig 4. Mesh View Simple Pipe

TABLE 4. Grid independency table

Grid Number	Nu	Nu Deviation %	f	f Deviation %
650170	95.5	2.1516	0.169	0.5917
890890	97.6	1.2145	0.168	0.8928
1075890	98.6	1.446	0.1665	0.5405
1197790	100.25	0.2686	0.1656	0.3623
1345900	100.52	-	0.165	-

As it can referred in above tabular representation grid independency test was conducted for CCR geometry. And all readings confer within standard range.

2.   Validation of Simple Pipe:-

For Numerical analysis selection of appropriate model is required. By using the different ansys models Nusselt Number(Nu) And Friction Factor (f) calculated and it further compared with the imperical value of Nu and f.(**Standard deviation range is 20%**)

TABLE 5. validation of simple pipe

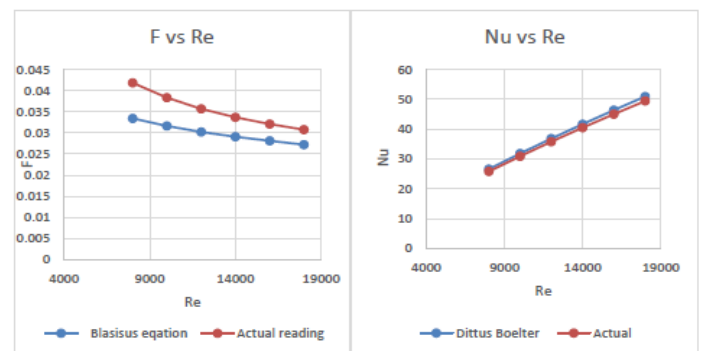| Re	f from blasiusEq	f from actual reading	Nu from dittus boelter Eq	Nu from actual Reading
8000	0.03341	0.04182	26.58	25.77
10000	0.03160	0.03831	31.79	30.83
12000	0.03019	0.03567	36.79	35.75
14000	0.02905	0.03368	41.63	40.44
16000	0.02810	0.03207	46.33	44.97
18000	0.02728	0.03073	50.91	49.38

Fig 5. Graph for f vs Re and Nu vs Re

3.   Numerical Analysis of pipe fitted with circular Ring inserts:-

Numerical Investigation of of the Heat Exchangers tube fitted with circular ring inserts have been carried out. The cases were analysed and Pitch to diameter ratio was taken as unity.

3.1.   Numerical analysis for plain pipe:-

The Numerical data is summarized in the following table-

TABLE 6. Readings from plain pipe

Re	V($ms-1$)	m(kg$s-1$)	Nu	f
8000	2.437	$5.515 \times 10-3$	25.772	0.0418
10000	3.046	$6.894 \times 10-3$	30.838	0.0383
12000	3.655	$8.274 \times 10-3$	35.754	0.0356
14000	4.265	$9.653 \times 10-3$	40.441	0.0336
16000	4.874	$11.031 \times 10-3$	44.975	0.0320
18000	5.484	$12.417 \times 10-3$	49.380	0.0307

Published by :

http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
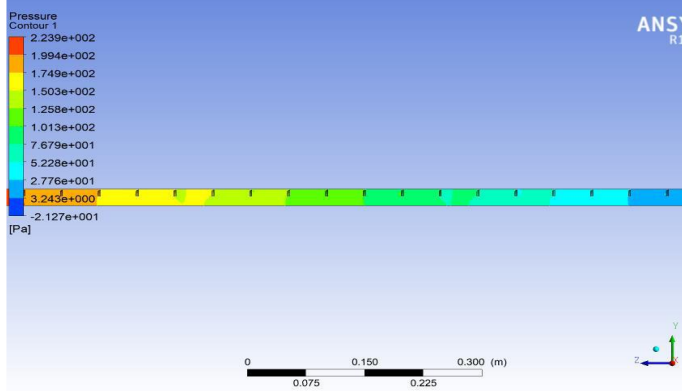**Vol. 10 Issue 10, October-2021**

4. CFD Analysis:-

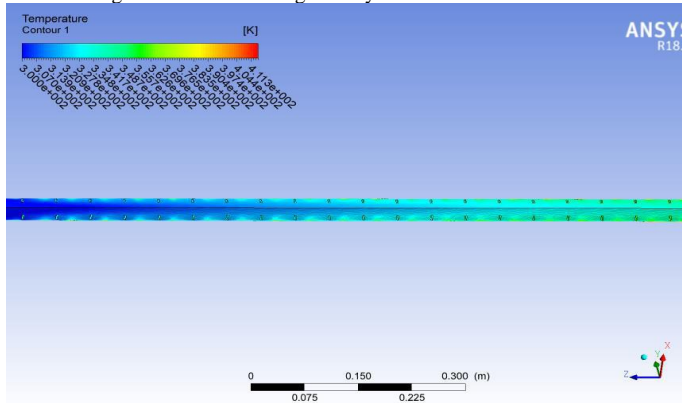Fig 6. Contour of CCR geometry 0.6 DR Case- Re 8000

Fig 8. Turbulence KE contour of tube with CCR (0.6 DR)

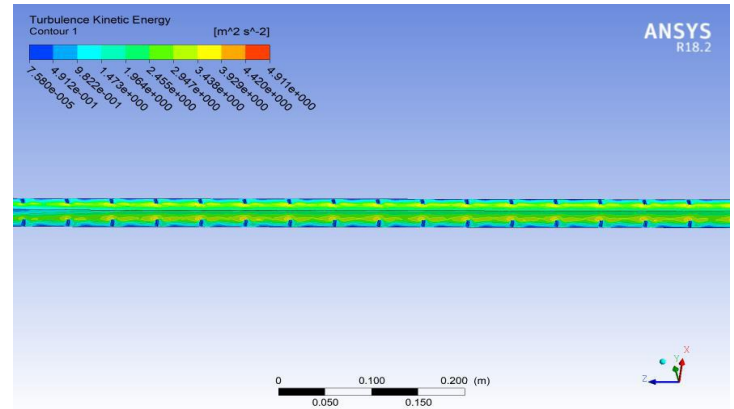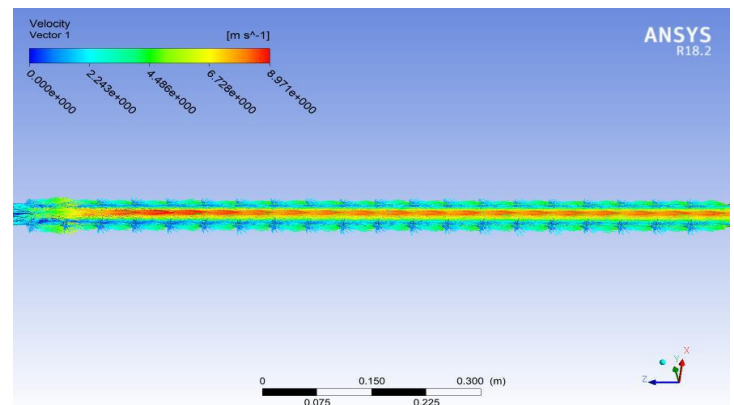Fig 7. Temperature contour of tube with CCR (0.6 DR)

Fig 9. Velocity Vector Contour of tube with CCR (0.6 DR)

5. Numerical Analysis of Ring Inserts With Conical Circular geometry:-
   5.1. Analysis of the tubes with CCR with DR 0.6, 0.7, 0.8 respectively:-
   Rings were inserted in such a way that there is Flow obstruction in the tube.
   The Numerical data obtained is summarized in the following table-

TABLE 7. Tube with CCR 0.6 DR

Re	$V(ms^{-1})$	$m(kgs^{-1})$	Ti (K)	To (K)	Tw (K)	$|\Delta P|$	Nu	F
8000	2.437	5.515×10^{-3}	300	341.66	345.68	223.10	107.263	1.6988
10000	3.046	6.894×10^{-3}	300	333.33	338.49	338.98	123.412	1.6303
12000	3.655	8.274×10^{-3}	300	327.77	333.56	479.69	137.936	1.5881
14000	4.265	9.653×10^{-3}	300	323.80	329.91	642.591	151.534	1.5502
16000	4.874	11.031×10^{-3}	300	320.83	327.03	829.315	164.766	1.5271
18000	5.484	12.417×10^{-3}	300	318.51	324.71	1041.71	177.775	1.5098

TABLE 8. Tube with CCR 0.7 DR

Re	$V(ms^{-1})$	$m(kgs^{-1})$	Ti(K)	To (K)	Tw (K)	$|\Delta P|$	Nu	f
8000	2.437	5.515×10^{-3}	300	341.76	352.12	106.781	85.5042	0.8131
10000	3.046	6.894×10^{-3}	300	333.41	343.93	161.410	99.1361	0.7764
12000	3.655	8.274×10^{-3}	300	327.84	338.27	227.699	111.673	0.7539
14000	4.265	9.653×10^{-3}	300	323.86	334.09	305.135	123.363	0.7375
16000	4.874	11.031×10^{-3}	300	320.87	330.86	393.986	134.359	0.7255
18000	5.484	12.417×10^{-3}	300	318.55	328.29	493.788	144.790	0.7157

TABLE 9. Tube with CCR 0.8 DR

Re	$V(ms^{-1})$	$m(kgs^{-1})$	Ti(K)	To (K)	Tw (K)	$|\Delta P|$	Nu	f
8000	2.437	5.515×10^{-3}	300	342.32	363.04	47.701	64.5984	0.3635
10000	3.046	6.894×10^{-3}	300	333.45	352.70	72.408	75.1583	0.3483
12000	3.655	8.274×10^{-3}	300	328.15	346.42	100.234	85.01	0.332
14000	4.265	9.653×10^{-3}	300	324.13	341.33	133.774	94.447	0.3235
16000	4.874	11.031×10^{-3}	300	321.11	327.39	172.309	103.419	0.3174
18000	5.484	12.417×10^{-3}	300	318.70	334.19	215.560	111.678	0.3125

5.2. Nusselt number ratio calculation:-
Nusselt number ratio is the ratio of Nusselt number for tube with insert to the Nusselt number for plain tube.

TABLE 10. Nusselt number ratio calculation

Re	For 0.6 DR [Nu/Nuo]	For 0.7 DR [Nu/Nuo]	For 0.8 DR [Nu/Nuo]
8000	4.1700	3.3177	2.5065
10000	4.0019	3.2147	2.4371
12000	3.8579	3.1233	2.3776
14000	3.7470	3.0504	2.3354
16000	3.6634	2.9873	2.2994
18000	3.6000	2.9321	2.2615

5.3. Friction factor ratio calculation:-
Friction factor ratio is the ratio of friction factor for tube with insert to the friction factor for plain tube

TABLE 11. Friction factor ratio calculation

Re	For 0.6 D[f/fo]	For 0.7 Dr [f/fo]	For 0.8 Dr [f/fo]
8000	40.6217	19.4428	8.6920
10000	42.5543	20.2657	9.0921
12000	44.6095	21.1684	9.3220
14000	46.0916	21.8953	9.6042
16000	47.5732	22.6012	9.8897
18000	49.1164	23.2823	10.1662

5.4. Thermal Performance Factor (n):-
Thermal performance factor is also called as overall enhancement ratio. TPF can be calculated as:-

TABLE 12. Thermal performance factor calculation

Re	For 0.6 DRTEF	For 0.7 DR TEF	For 0.8DR TEF
8000	1.2107	1.2338	1.2190
10000	1.1463	1.1700	1.1678
12000	1.0886	1.1297	1.1300
14000	1.0450	1.0904	1.0987
16000	1.0107	1.0562	1.0709
18000	0.9830	1.0268	1.0439

VII.   RESULTS AND DISCUSSION:-

The heat transfer and fluid flow characteristics were studied using the numerical simulation of fluid flow. The observation that can be made is; higher heat transfer rate is obtained in test section with ring blockages as compare to the smooth tube
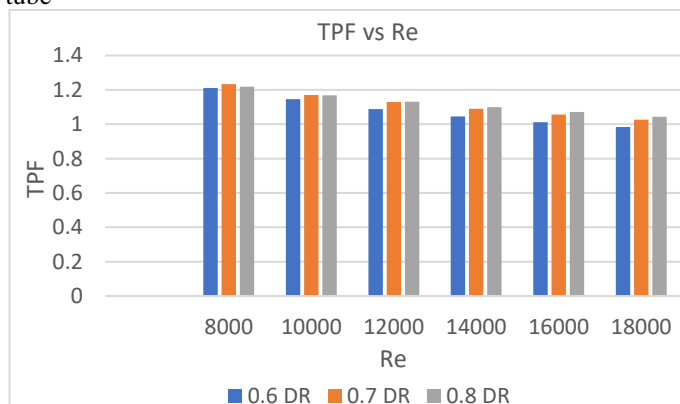

Fig 10. Graph for *TPF vs Re*

VIII.   CONCLUSION:-

1. As per result the Nu and f both were increasing in the tube with insert as comparing with simple pipe.
2. As observed in result and discussion Nusselt number increases with decrease in friction factor.
3. Thermal performance factor is maximum for 8k RE CCR and minimum for 18k RE CCR.
4. The maximum thermal performance factor of 1.2338 is found by the use of the CCR type ring inserts with 0.7DR

IX.   REFERENCES:-

[1] C.B. Sobhan , K.T. Mohammed kutty, M. Hannan, and P. Krishtaiah, "Experimental investigation on a 1-2 heat exchanger with wire-wound tubes," warme- und Stoffubertragung, vol. 29, March.1994, pp. 211-217.

[2] Cengiz Yildiz, Yasar Bicer, and Dursun Pehlivan, "Effect of Twisted Strips on Heat Transfer and Pressure Drop in the Heat Exchangers," Energy Conversion and Management, vol. 39, March.1998,pp. 331-336.

[3] Kenan Yakut, and Bayram Sahin, "Flow-induced vibration analysisof conical rings used for heat transfer enhancement in heat exchangers," Applied Energy vol. 78, Sept. 2004, pp. 273-288, doi:10.1016/j.apenergy.2003.09.001.

[4] J. U. Ahamed, S. Ahmed, Y. Nukman, M. A. Wazed, T. M. Y. S. Tuan Ya, Enhancement and Prediction of Heat Transfer Rate in Turbulent Flow through Tube with Perforated Twisted Tape Inserts, Journal of Heat Transfer April 2011, Vol. 133.

[5] Aydin Durmus, "Heat transfer and exergy loss in cut out conical turbulators," Energy Conversion and Management vol. 45, March. 2004, pp. 785-796, doi:10.1016/S0196-8904(03)00186-9.

[6] K. Yongsiri , P.Eiamsa-ard , K.Wongcharee , S.Eiamsa-ard, Augmented heat transfer in a turbulent channel flow with inclined detached-ribs, Case Studies in Thermal Engineering 3(2014)1–10.