CFD based performance analysis of a roughened solar air heater duct having NACA 0030 airfoils as artificial roughness

Rohan Sonalkar[1], Parth Trivedi[2]

[1,2]Students, Dept. of Mechanical Engineering, SVIT Vasad, Gujarat, India

---

Abstract – *The heat transfer rate in a solar air heater duct has been found to be low due to the formation of laminar sub-layer when air passes through the absorber plate. The use of artificial roughness on the underside of the absorber plate is an effective technique to break this laminar sub-layer and hence to enhance the rate of heat transfer. In the present work, Computational Fluid Dynamics (CFD) is used to analyze the air-flow and heat transfer rate in the duct of solar air heater. RNG k-epsilon turbulence model is selected by comparing the results obtained from different turbulence models. A detailed numerical analysis is carried out using this model, firstly, for a smooth plate and then with a series of NACA 0030 airfoils as artificial roughness arranged with an appropriate pitch on the absorber plate for Reynolds number ranging from 6000 to 18000. A constant solar radiation of 1000 W/m² is provided on the absorber plate. Correlations for Nusselt number and friction factor have been developed which show a significant increase in the heat transfer rate in comparison to that for a smooth surface without noticeable friction losses.*

***Key Words*: Solar air heater, Heat transfer rate, Artificial roughness, NACA airfoil, Friction factor, CFD analysis**

1. INTRODUCTION

1.1 Energy

Energy in its various forms has played a significant role in global economic progress and industrialization. As the world's fossil fuel reserves are depleting rapidly, the development of non-conventional renewable energy sources has caught attention. Sunlight is a direct and perennial source of energy which provides a non-polluting reservoir of fuel. The simplest and most efficient method to utilize this freely available energy is to transform it into thermal energy for heating applications by using solar collectors. One of the most widely used collection device is a solar air heater. A solar air heater is preferred due to minimal use of materials in it and its cost. The major applications of solar air heaters are space heating, curing of industrial products and seasoning of timber [1-5]. Figure -1 shows a conventional solar air heater which is considered for analysis. Design and construction details of such type of conventional system are described by Garg and Prakash [6].

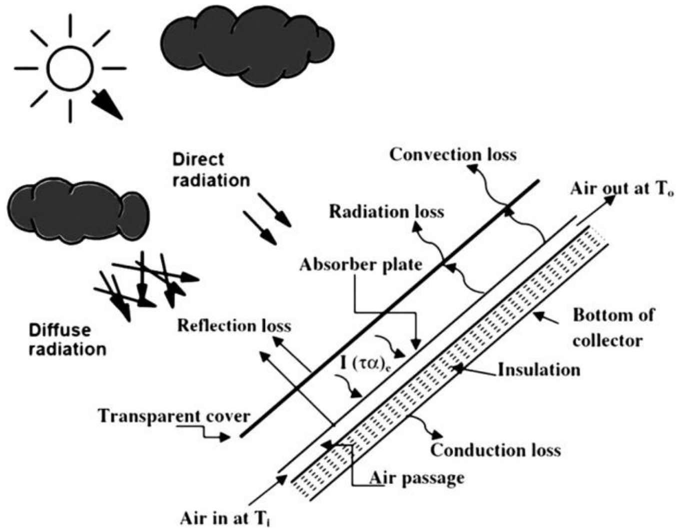

Figure -1: Conventional solar air heater

1.2 Thermal efficiency of a solar air heater

The thermal efficiency of solar air heaters has been found to be poor because of their inherently low heat transfer capability between the air flowing in the duct and the absorber plate. There are two basic techniques for improving the heat transfer co-efficient between the absorber plate and air. The first method involves the use of extended surfaces called fins to increase the area of heat transfer without affecting the convective heat transfer co-efficient. The second way is to increase the convective heat transfer by creating turbulence at the heat transferring surface. This can be done by providing artificial roughness on the underside of the absorber plate. Various investigators have attempted to design a roughness element, which can increase the heat transfer without significant frictional losses.

Hans et al. [7] studied the effect of multiple V-shaped roughness on heat transfer. There was an increase in heat transfer with increase in relative roughness height and decrease in relative roughness pitch. Alam et al. [8] carried out experiments to determine the effect of non-circular perforation holes and circular holes in V-shaped ribs attached to the absorber plate. Higher nusselt number was obtained for non-circular holes. Saini and Verma [9] studied the effect of roughness on the performance of solar air heater duct provided with dimple shaped roughness geometry.

Research studies on artificial roughness using CFD techniques are less numerous. However, Lee [10] attempted to study heat transfer and air-flow in a duct provided with two-dimensional roughness using a CFD model. Chaube [11] performed a numerical analysis using FLUENT 6.1 to analyze the effect of nine types of roughness on heat transfer enhancement and friction characteristics.

In the present work, roughness element in the form of NACA 0030 airfoil has been used. The heat transfer and flow analysis for the chosen r oughness geometry has been carried out using 3-D models.

1.3 NACA airfoils

The NACA airfoils are airfoil shapes for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). The shape of NACA airfoils is described using a series of digits following the word "NACA". The parameters in the numerical code can be entered into equations to precisely generate the cross-section of the airfoil and calculate its properties. The NACA 0030 airfoil is symmetrical, the 00 indicating that it has no camber. The 30 indicates that the airfoil has a 30% thickness to chord length ratio: it is 30% as thick as it is long. Figure -2 shows the 3-dimensional NACA 0030 airfoil used in the analysis, whereas table -1 show the coordinates used to plot the profile.

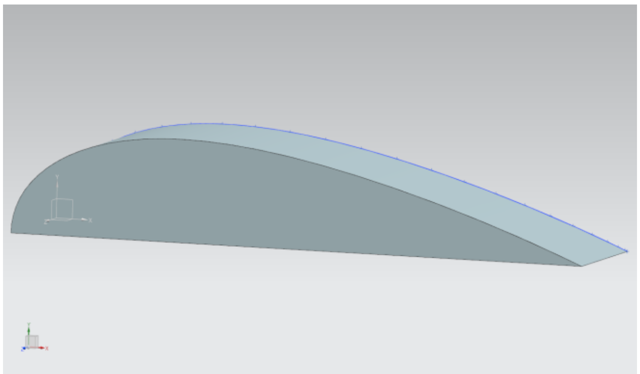

Figure -2: NACA 0030 airfoil

100	**0.315**	**0**
99.6057	0.453	0
98.4292	0.8607	0
96.4888	1.5205	0
93.8153	2.4048	0
90.4508	3.4786	0
86.4484	4.7024	0
81.8712	6.0338	0
76.7913	7.4292	0
71.289	8.8434	0
65.4508	10.2294	0
59.3691	11.5372	0
53.1395	12.7136	0
46.8605	13.7025	0
40.6309	14.4473	0
34.5492	14.8937	0
28.711	14.995	0
23.2087	14.7166	0
18.1288	14.0401	0
13.5516	12.9656	0
9.5492	11.5122	0
6.1847	9.7146	0
3.5112	7.6182	0
1.5708	5.272	0
0.3943	2.721	0

Table -1: Coordinates used to plot the profile

2. DETAILS OF SOLAR AIR HEATER DUCT

As per ASHRAE 93-77 [12] recommendations, the system and operating parameters have been considered for the present investigation. The most important part of the system considered was the duct which was considered having inner cross section dimensions of 300mm*25mm as shown in the figure -3. The aspect ratio has been kept 12 in this study, as many investigators have established this aspect ratio for such studies. The flow system consists of 900mm long entry section, 1000mm long test section and 500mm long exit section. Entry and exit length of the flow have been kept to reduce the end effects on the test section considering the recommendation provided in ASHRAE standard 93-77 [12]. A constant heat flux of 1000 W/m^2 was considered to be supplied by having a heater plate placed over the absorber plate.

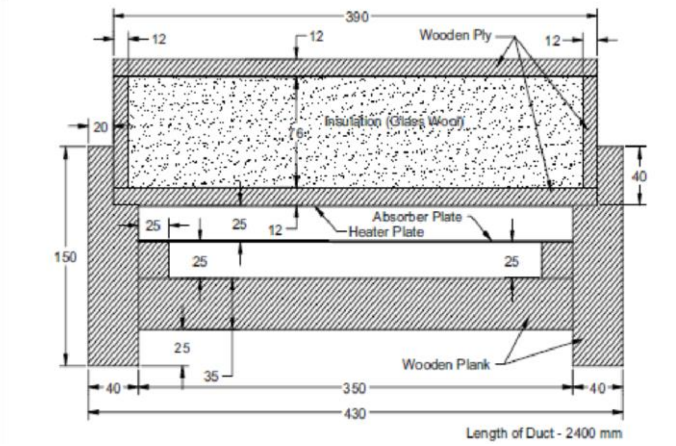

Figure -3: Sectional view of solar air heater duct

3. ANALYSIS

3.1 Solution domain

3.1.1 General assumptions

1. The flow is steady, incompressible.
2. The thermal conductivity of the duct wall does not change with temperature.
3. The duct wall material is homogeneous and isotropic.
4. The working fluid is air, which is assumed to be incompressible.

3.1.2 Solution technique

The arrangement of roughness elements in the form of NACA 0030 airfoils arranged on the inner side of the absorber plate has been considered along with flat duct. The solution domain used for CFD analysis has been generated as shown in figure -4. The duct used for analysis is having the height (H) of 25 mm and width (W) of 300 mm. Thickness of the absorber plate has been considered as 0.5 mm. A 28 mm thick wooden plank was considered for the sides of the duct and 40 mm thick wooden plank as bottom of the duct. The mean inlet velocity is calculated using Reynolds number, velocity inlet is used as inlet boundary condition and outflow is used as outlet boundary condition. A uniform heat flux of 1000 W/m^2 is considered for analysis to the test wall, whereas all other walls are considered to be insulated. Second order upwind Numerical scheme and simple algorithm are utilized to discretize the governing equations.

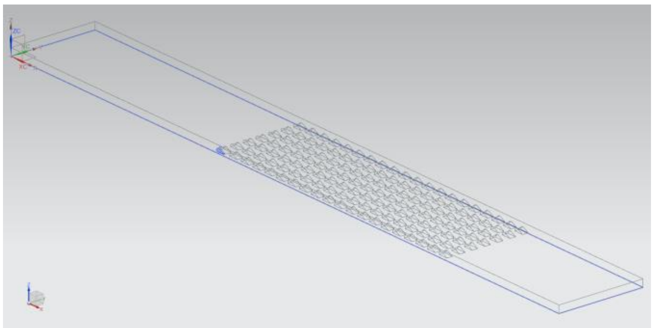

Figure -4: Arrangement of NACA 0030 airfoils

3.2 Grid

3.2.1 Grid generation

The chosen roughness geometry is such that secondary flows would definitely occur. So, the possibility of using 2-D solution domain and grid is ruled out. Thus, 3-D solution domain and grid were selected. In order to examine the flow and heat transfer critically in the inter airfoil regions, finer meshing at these locations has been done. In other regions coarser mesh has been used. For the present work, meshing has been done using commercially available software ANSYS FLUENT 16.2. The mesh generated is shown in the figure -5.

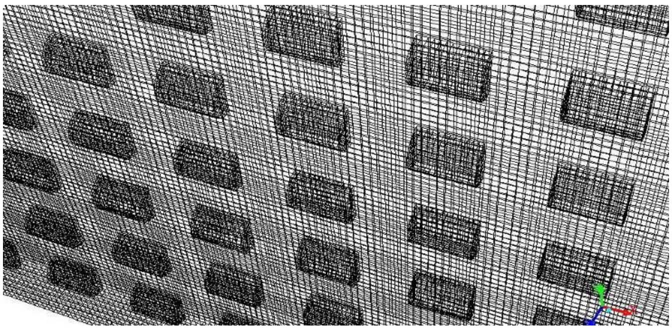

Figure -5: Mesh generation

3.2.2 Grid independence test

To study the effects of grid, number of grid elements were varied from 60,000 to 15,00,000 in various steps. It was found that after 13,78,000 cells, further increase in cells has less than 1% variation in Nusselt number value as shown in figure -6. Hence further analysis is done with 13,78,000 elements.

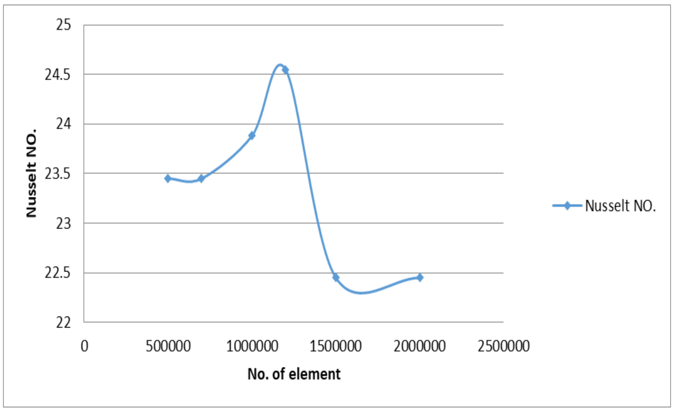

Figure -4: Grid independence test

3.3 CFD Analysis

3.3.1 Selection and validation of turbulence model

The values obtained by using various turbulence models were compared with the experimental results of Alam et al. [8] and with Dittus-Boelter correlation for nusselt number (Nu) and modified blasius equation for friction factor (f).

$$Nu = 0.023\, Re^{0.8}\, Pr^{0.4}$$

$$f = 0.085\, Re^{-0.25}$$

The RNG k-epsilon model gave the closest values and hence it was used as the turbulence model along with energy model.

3.3.2 Boundary conditions

The boundary conditions were kept as shown in the table -2.

Inlet type	Velocity inlet
Outlet type	Pressure outlet
Duct bottom	Heat flux (1000 W/m^2)
Other surfaces	wall
Velocity	Varies with number Reynolds ranging from 6000 to 18000

Table -2: Boundary conditions

4. RESULTS AND DISCUSSIONS

We carried out analysis on ANSYS FLUENT 16.2 for two conditions:

1. Analysis of flat duct without any roughness at various velocities.
2. Analysis of duct with series of NACA 0030 airfoils as artificial roughness at various velocities.

For both, velocities were calculated from Reynolds number ranging from 6000 to 18000 and hydraulic mean diameter.

4.1 Result tables

Table -3: Result table for flat duct

Reynolds number	velocity	Inlet (T)	Outlet (T)	Nusselt number	Friction factor
6000	1.89	300	343.39	21.27	0.010417
8000	2.53	300	330.55	26.32	0.009049
10000	3.16	300	327.19	30.02	0.008228
12000	3.79	300	321.97	33.85	0.008089
14000	4.43	300	318.92	40.58	0.007941
16000	5.06	300	316.69	44.32	0.007558
18000	5.69	300	312.97	47.87	0.007223

Table -4: Result table for duct with NACA 0030 airfoils

Reynolds number	velocity	Inlet (T)	Outlet (T)	Nusselt number	Friction factor
6000	1.89	300	342.4	18.33	0.009608
8000	2.53	300	331.55	22.51	0.008001
10000	3.16	300	325.2	27.45	0.007108
12000	3.79	300	320	31.25	0.006337
14000	4.43	300	317.92	35.08	0.005597
16000	5.06	300	315.69	39.03	0.005107
18000	5.69	300	313.94	41.03	0.004856

4.2 Graphs

Graphs of Nusselt number vs Reynolds number and friction factor vs Reynolds number were plotted for both flact duct and duct with NACA 0030 airfoils.

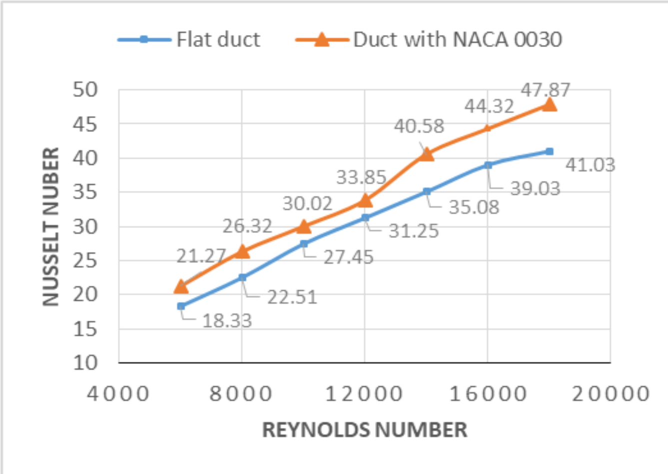

Figure -8: Nusselt number vs Reynolds number

Figure -8 shows the graph of Nusselt number vs Reynolds number for both cases. The blue line depicts the values of nusselt number for various Reynolds number for a flat duct, whereas the red line shows the same values for duct with NACA 0030 airfoils. It is evident that with increase in Reynolds number the Nusselt number also increases in both cases. However, the values of Nusselt number for a particular Reynolds number is much higher for duct with NACA 0030 airfoils than without them. This clearly shows enhancement of heat transfer due to provision of artificial roughness. Also the highest value of Nusselt number is obtained at highest value of Reynolds number taken i.e. at 18000.

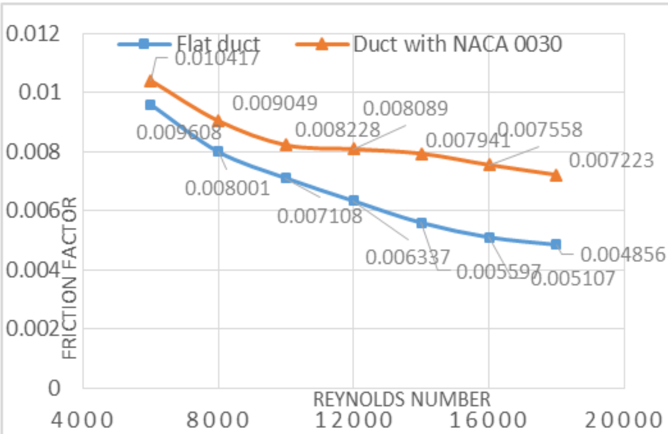

Figure -9: Friction factor vs Reynolds number

Figure -9 shows the graph of Friction factor vs Reynolds number for both cases. The blue line depicts the values of friction factor for various Reynolds number for a flat duct, whereas the red line shows the same values for duct with NACA 0030 airfoils. It is evident that with increase in Reynolds number the friction factor decreases in both cases. However, the values of Friction factor for a particular Reynolds number is much higher for duct with NACA 0030 airfoils than without them. This shows some increase in frictional losses due to provision of artificial roughness. Also the lowest value of friction factor is obtained at highest value of Reynolds number taken i.e. at 18000.

4.3 Contours

We applied velocity as inlet condition. The velocity of air flowing through the duct decreases due to friction which is shown by figure -10.

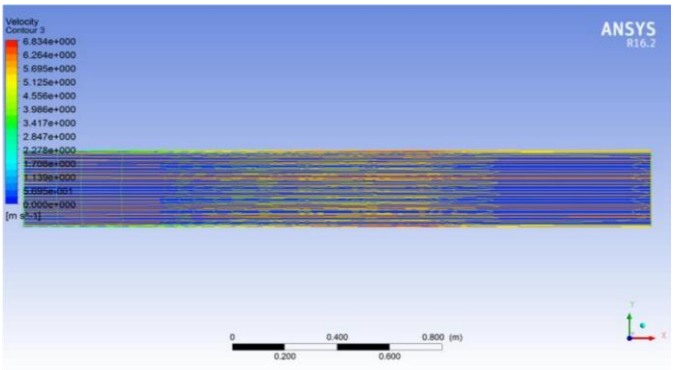

Figure -10: Velocity contour

There is temperature variation across the duct from top surface to bottom surface which is shown by figure -11.

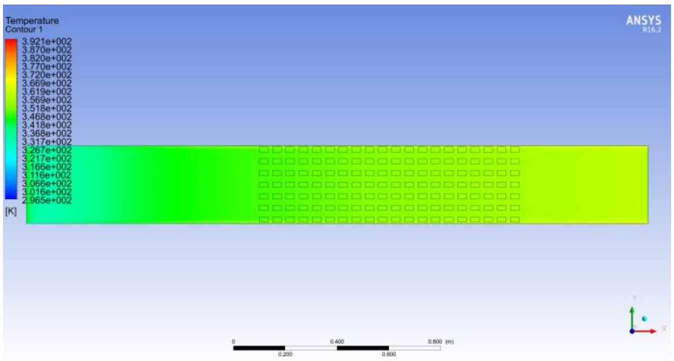

Figure -11: Temperature contour

There is a disturbance in turbulent energy due wall's solid particles. Its variation across the duct is shown by figure -12.

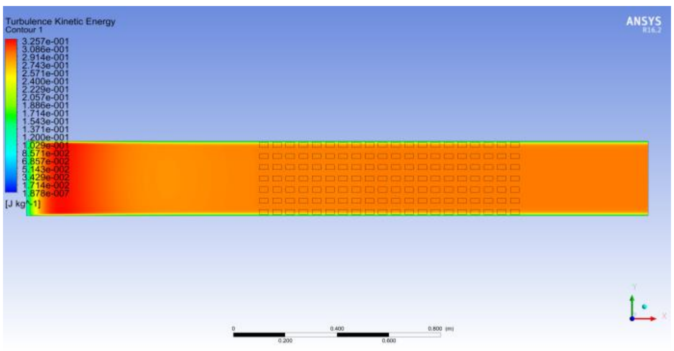

Figure -11: Turbulent kinetic energy contour

All the contours shown above are for duct provide with NACA 0030 airfoils and for Reynolds number of 18000.

5. CONCLUSIONS

In this paper, we have presented the results obtained by numerical analysis of air-flow and heat transfer in the duct of a solar air heater, whose absorber plate is provided with artificial roughness in the form of NACA 0030 airfoils. The analysis is based on CFD techniques and was performed using numerical software ANSYS FLUENT 16.2. This numerical analysis allowed us to found out the effect of roughness on the air flow and heat transfer enhancement in solar air heaters.

The first part of the analysis aims to validate the turbulence model by comparing different results. We found RNG k-epsilon model the best and hence used it for analysis. The second part is an approach to analyze the thermal performance of solar air heater in real operating conditions. We have plotted Nusselt number and friction factor variations with Reynolds number. The results are indicators of the effect of NACA 0030 airfoils in the heat transfer enhancement in solar air heater.

Moreover, the geometric shape of roughness of NACA 0030 airfoils gave rise to heat transfer with moderate friction and therefore not penalizing the thermal performance.

So, we recommend the use this type of roughness to improve the thermal performance of solar air heater.

REFERENCES

[1] Foster R, Ghassemi M, Cota A. Solar energy: renewable energy and the environment. New York: CRC Press Taylor & Francis group; 2010.

[2] Li X. Green energy: basic concepts and fundamentals. 1st ed.. New York: Springer; 2011.

[3] Maczulak A. Renewable energy: sources and methods. 1st ed.. New York: Infobase Publishing; 2011.

[4] Kaltschmitt M, Streicher W, Wiese A. Renewable energy: technology, eco- nomics and environment. 1st ed.. New York: Springer; 2007.

[5] Quaschning V. Understanding renewable energy systems. 3rd ed.. London: Earthscan; 2005.

[6] Garg HP, Prakash J. Solar energy fundamentals and applications.1st ed.. New Delhi:TatabMcGraw-Hill

[7] V. S. Hans, R.P. Saini b, J.S. Saini Performance of artificially roughened solar air heaters—A review, Renewable and Sustainable Energy Reviews 13 (2009) 1854–1869.

[8] Alam Tabish, Saini Rp Saini JS Experimental investigation on heat transfer en- hancement due to V-shaped perforated blocks in a rectangular duct of solar air heater. Energy conservation and Management 2014; 81:374-383.

[9] Saini RP, Verma J. Heat transfer and friction factor correlations for a duct having dimple-shaped articial roughness for solar air heaters.Energy 2008;33:127787.

[10] Lee C.K., S.A. Abdel-moneim Computational analysis of heat transfer in turbulent flow past a horizontal surface with two-dimensional ribs int. Comm. Heat Mass Transfer, 2001,Vol. 28, No. 2. pp. 161-170

[11] Chaube A, Sahoo PK, Solanki SC. Analysis of heat transfer augmentation and flow characteristics of a solar air heater. Renew Energy 2006;31:317–31.

[12] ASHRAE Standard 93-77. Methods of testing to determine the thermal performance of collectors. Atlanta, GA: American Society of Heating, Refrigerating and Air-conditioning Engineers Inc; 1977.

BIOGRAPHIES

ROHAN SONALKAR is a final year student, pursuing B.E. MECHANICAL from SARDAR VALLABHBHAI PATEL INSTITUTE OF TECHNOLOGY, VASAD affiliated to GUJARAT TECHNOLOGICAL UNIVERSITY (GTU), GUJARAT, INDIA

PARTH TRIVEDI is a final year student, pursuing B.E. MECHANICAL from SARDAR VALLABHBHAI PATEL INSTITUTE OF TECHNOLOGY, VASAD affiliated to GUJARAT TECHNOLOGICAL UNIVERSITY (GTU), GUJARAT, INDIA