International Journal of Engineering Technology & Advanced Learning
ISSN: 3107-4227, Vol. 01 (01), March 2025, pp. 27-34

Investigation of Heat Transfer and Fluid Behavior in Radial Microchannel Heat Sinks

Saif Ali Ansari[1], Sachin Baraskar[1]

[1]*Department of Mechanical Engineering, SSSUTMS University, Sehore,, Madhya Pradesh, India.*

Abstract

Due to miniaturization the concentration of the ICs has increased multifold on the motherboard. With the increase in the concentration of ICs, heat dissipation from microelectronics equipments is also an associated area of concern. More and more work has started on the thermal management of the electronics equipment to make them efficient, reliable, and stable by modifying the designs of the heat sinks that are used to dissipate the heat from the electronics product. Recently the high speed integrated circuits are in high demand in different segments of the engineering like Aeronautical Engineering, Electronics Engineering, Automotive Engineering Consumer and Medical fields that generate high heat flux which in turn needs to be dissipated and for that reason sophisticated technologies to cool the substrate started evolving.

*Keywords***:** *Micro electric framework, ANSYS, Convection, Cooling..*

1. Introduction

Radial micro channels give the maximum surface area for departure of the thermal energy from the exterior surface of device. The different combinations for heat sink having micro channels aligned radially give various results of heat transfer. In the spiraling micro channel the stream pace of liquid was assumed an essential function in the thermal expulsion. The impelled convection of liquid additionally improves the rate of thermal expulsion when contrasted with the whole some manoeuvre of the liquid finished with the assistance of structural framework. The different states of inlet and outlet plenum likewise have the variable impact on heat expulsion rate.

1.1. Type of fluid flow in microchannel

It has been observe dthat there is significant contrast between the boundary layer conditions for both type of flows (external flow geometry and the internal flow geometry) [1]. In the internal flow, the boundaries of the surface confined the fluid in itself and in the external flow the fluid does not obey any boundary conditions. The region from where the fluid enters is called entrance region. The flow of the fluid is laminar or turbulent is totally independent of the entrance region. Reynold number is the criteria to decide the flow to be laminar or turbulent. If the value of Reynolds number exceeds 2300 the flow turns out to be turbulent. Heat transfer is dependent entity of fluid flow and the boundary conditions.

2. Literature Review

Qu &Mudawar (2022) [38] this paper illustrates about the numerical and experimental study done mainly on the drop in pressure and the heat exchange through heat sink comprising microchannels in solitary stage flow system. Deoxygenated copper having PCP plate fitted on top to cover was used for the creation of microchannel heat sink. Microchannels of rectangular shape with 231 µm width and depth of 713 µm were arranged on the heat sink for coolant, Deionized water was utilized. The test module comprised of microchannel heat sink, lodging, plate for covering at top and twelve cartridge heat radiators. The three directional thermal exchange attributes of the heat sink were examined mathematically. Additionally introduced and examined is an itemized portrayal of the locale and usual heat exchange qualities of heat sink. The deliberate drop in pressure and temperature distribution show great concurrence as compared with mathematical expectations.

Steinke & Kandlikar (2022) [39] in this paper heat transfer in single phase liquids was discussed in case of plain and enhanced microchannels. An Enhanced microchannel heat exchanger (EMCHX) that utilized single stage fluid streams was created. A silicon surface was chosen in order to fabricate the fins in form of strips in microchannel flow area. The enhanced microchannels provide a higher coefficient of performance even when flow conditions remain same. The liquid admitted flow section and goes in the flow bearing a small separation. The liquid experiences a flow block by virtue of another blade. The liquid must separate and move around the following blade. The whole cycle was

E-mail ID: saifali121297@gmail.com

rehashed as the liquid descended the whole stream length. Accordingly, the liquid never went into a completely evolved state and should divide and move around the following fins. This gadget would utilize the enhancement found in developing streams.

Moreover, the subsequent blending will constrain the liquid to have more communication with the warmed dividers and furthermore give heat exchange improvement. The honed edges gave enough free regions so as to permit the liquid to pass the fins. Silicon microchannels enhanced heat transfer as compared with the plain and continuous wall microchannels. The

Comparison results in drop in pressure across microchannel test section. For experimentation upgraded microchannel heat exchanger was manufactured and tried. The outcomes showed improvement in thermal potential for the general thermal obstruction of upgraded microchannel heat exchanger contrasted with one utilizing plain microchannels. Though having increased drop in pressure but enhanced microchannel showed better performance with COP of the level of 290 as compared with normal microchannel which had COP of 11.

Özdemir (2018) [40] this research paper states a brief review on the single phase and two stage drop in pressure parameters and fluid current irregularities in microchannels. This investigation indicated that the continuum hypothesis was material for single stage drop in pressure applications with certain contemplations. Likewise, the two stage drop in pressure in microscale was having comparative qualities with traditional scale micro channels. The impact of mass flux, heat transition, test conditions, channel mathematical boundaries like hydraulic diameter, aspect ratio was governed thoroughly over a wide scope of writing examines including the past and ongoing investigates. Also, traditional and miniature size pressure drop relationships are talked about. Trials were done on rectangular horizontal with Dh = 0.25 mm - 0.99 mm in laminar and turbulent stream systems utilizing water. It's been finished up here that the traditional scale hypothesis can foresee well the pressure drop information in miniature size if the vulnerabilities and passage/exit diminutions are deliberately thought of. In another examination has been led by Xu et al. examined aluminum and silicon microchannels having Dh = 0.03-0.39 mm in laminar and turbulent stream systems representing passageway and exit diminution. The traditional completely created flow hypothesis could all the more likely anticipate the friction factor consequences of silicon microchannels. The creators credited this to the high estimation mistakes in aluminum microchannels due to machining. Then again, silicon microchannels were bonded to the wafer. Hereafter, channel measurements of silicon microchannels were estimated precisely.

Weilin Qu 2002 [41] in this paper three dimensional heat transfer analysis was done experimentally in microchannel heat sinks. Water was taken as coolant in Rectangular microchannel heat sink fabricated from silicon wafer of 1 sq cm. Microchannels 57 µm wide with profundity of 180 µm were isolated by 43µm divider. A mathematical codes dependent on the finite diff erence strategy and SIMPLE algorithm was created to understand the controlling conditions. It‘s inborn favorable circumstances pulled in significant considerations from the code which were approved by contrasting the forecasts and explanatory solutions with accessible experimental information. On being examined, temperature ascended approximately linear alongside with stream course in compact and in fluid locales. Noteworthy temperature rise was experienced at bottom of heat sink quickly over the channel exit. The heat flux and Nusselt number had greater magnitude close to micro channel bay and fluctuate around micro channel outskirts, dip towards zero in the corners

Qu &Mudawar (2002a) [38] this paper presents a trial investigation of heat exchange and drop in pressure attributes of solitary stage microchannel cooling framework with spiraling radial incursion of coolant for high heat flux. The framework was heated from one conducting partition made of copper and for working fluid, water was used. The microchannel had 1 cm arch and 300 microns vertical gaps. Trials showed an average of 76% bigger drop in pressure contrasted with expository model for laminar stream in parallel disk which was having radially spiraling inflow. The normal variability in temperature of substrate was 18% of the bulk liquid temperature gain over the gadget. The framework indicated promising cooling attributes for gadgets and with heat flow of 113 W/cm2 at a surface temperature of 77°C.

Al Siyabi et al. (2018) [42] this paper gives mathematical examination along with experimentation. Study was made on multi incrusted micro channel heat sink being used on concentrating photograph voltaic application. A multi incrusted microchannel heat sink strategy was viewed as comparatively efficient as far as thermal expulsion and siphoning power in contrast to numerous other procedures for lowering the temperature. COMSOL Multi-physics was utilized to reenact the model utilizing the conditions of managing mass, force and dynamism for single stage laminar flow. The thermal efficiency was improvised significantly by increasing the quantity of layers from single to triple layers by 20%.

Chai et al. (2013) [43] this paper had test examination on heat exchange improvement of microchannel heat sinks with intermittent enlargement and squeezing of cross segments is explored both by trial run and mathematically. Fabrication of heat sink was done in such a way that it was having 10 equidistant microchannels with width of 0.1 mm and depth of 0.2 mm down consistent area and each microchannel comprises of a variability of intermittent enlargement and squeezing cross section. Three dimensional laminar mathematical simulations, in light of the Navier Stokes conditions and energy condition, were acquired for drop in pressure and thermal exchange. The impacts of intermittent expanding and squeezing cross segments on pressure drop, heat exchange and heat obstruction was examined individually. The impacts of the entry and departure plenum area and the lateral pieces of silicon wafer on liquid stream and heat exchange was examined.

Literature suggest most of the work related to single phase fluid flow on parallel rectangular shaped micro-channels and to the best knowledge of the author, very less data is available on Heat exchange and stream properties of fluid in the radial micro channel heat sink (RMCHS). The RMCHS have less pressure drop between inlet and outlet plenum and more uniform temperature gradient in comparison to the parallel flow micro channel. Flow inlet and outlet conditions of micro-channel (RMCHS) have to be taken in to account in depth for efficient design of micro-channels. It is also generic that the circular shape micro channel has the maximum surface area for heat dissipation. As on today flow and heat transfer analysis of the macro channels has be extended to micro-channels, but not much works have been initiated to validate these correlations on the RMCHS. Because of the different shapes of the plenum and inlet/outlet plenum size, the entrance and exit arrangement of fluid flow will definitely have the significant effect on the heat exchange characteristics of the RMCHS. Micro-channels are fabricated with different dimensions to meet the requirement of a particular application. Different flow arrangements of the coolant and its effect on the heat transfer rate needs to be studied elaborately. Plenum shape may also have the effect on the flow of coolant hence on the heat transfer rate, thus this is also be the area of interest for the work. Generally thermo-physical properties of liquids vary with temperature. After identifying the specific problem in micro-channel cooling, the following objectives can be briefed.

3. Methodology

The rectangular micro channel having a fixed length of 35 mm and its width varies from 0.8 mm to1 mm along with the height which varies from 3.2 mm to 4 mm. In the present study the results of the 0.8 mm width with 3.2 mm height are considered making an aspect ratio as 4, similarly for the aspect ratio 5 and 8. Height of the channel is kept same but the width varies. Study also extended by varying width of square inlet plenum with side 10, 20 and 30 mm to observe the effect of width variation on various output parameters of the radial micro channel heat sinks presented in Figure 5.1. The radial micro channel has varying diameter of circular inlet plenum with diameter10, 20 and 30 mm as shown in Figure 5.2. One-dimensional flow of fluid is considered in the radial direction. At the entrance of the heat sink, constant average temperature of 300K was kept. Here heat transfer is a conjugate problem, a combination of heat transfer through conduction and exchange of heat through convection. Heat is transferred by conduction through solid and dissipated away by cooling fluid by convection. Figure 5.3 shows the orthographic view.

Figure 1. 2D and 3D drawing of rectangular micro channel (A-Front view, B-Top View, C&D- Isometric View).

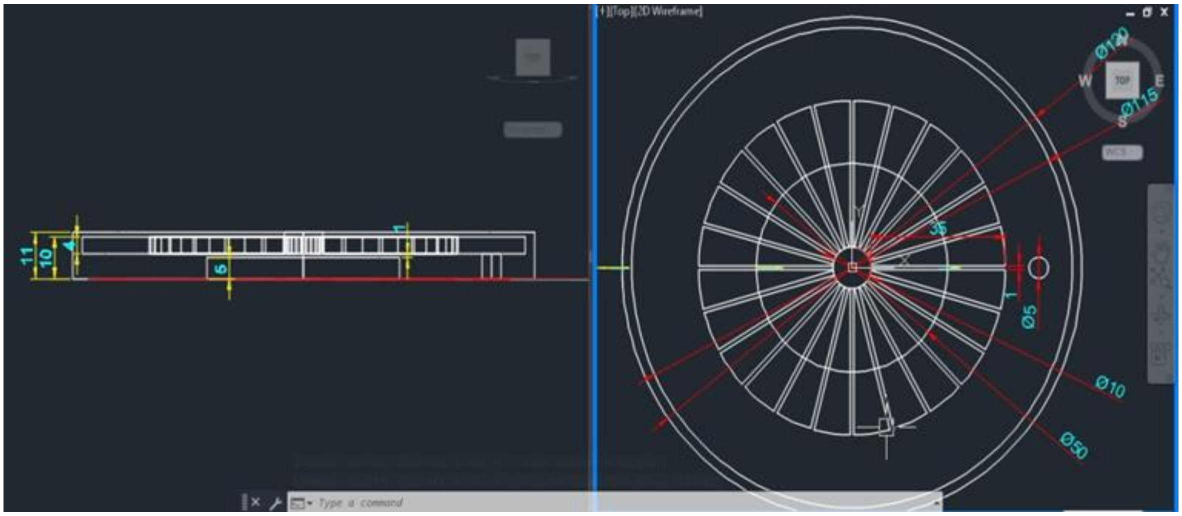

Figure 2. 2D drawing of Circular inlet radial micro Channel (Front View and Top view)

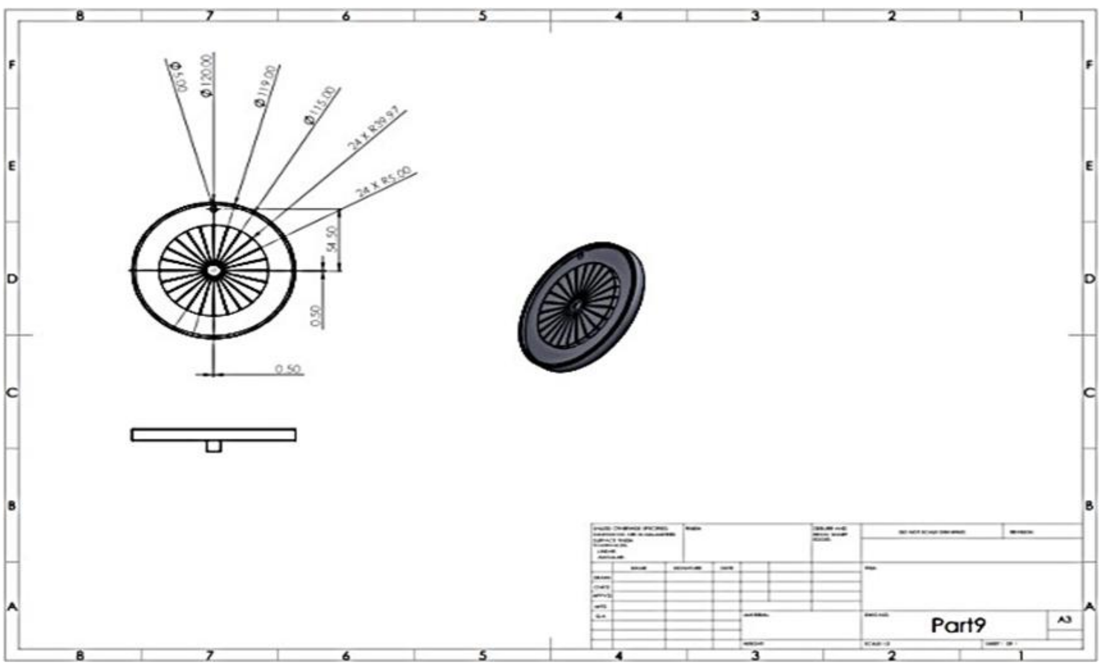

Figure 3. 2-Dimentional and 3-Dimentional drawing of radial micro channel.

3.1. Properties of Micro Channel Materials

Industrial copper is taken as material of the sink. The characteristics of material listed in following Table 1.

Table 1: Characteristics of copper

Property	Value	Unit (Metric)
Density	8.92	g/cm^3
Meltingpoint	1083	^oC
Boilingpoint	2595	^oC

Property	Value	Unit (Metric)
Latentheatof fusion	205	J/g
Thermal conductivity at 20ºC	3.94	W/(m.k.)
Modulus of elasticity	118000	MPa
Modulus of rigidity	44000	MPa

3.2. ANSYS FLUENT

Fluent has been used as solver for numerical experimentation work. Three popular solvers are available in ANSYS FLUENT. These are given below-

(i) Pressure based segregated solver (PBSS)
(ii) Pressure based coupled solver (PBCS)
(iii) Density based coupled solver(DBCS)

As per the ANSYS manual, among above three, first one (PBSS) has been used due to following reasons –

(i) It is suitable for incompressible fluid as assumption is made.
(ii) It solves three momentum equations, continuity and energy equations. It updates velocity also.
(iii) Pressure based segregated solver is applicable to wide variety of problems.
(iv) It requires less memory or lower RAM requirements.
(v) This solver is also quick than other solvers.

Though analysis could be done in 2D and 1D in ANSYS but in this study volumetric analysis is to be done therefore 3D modelling is done.

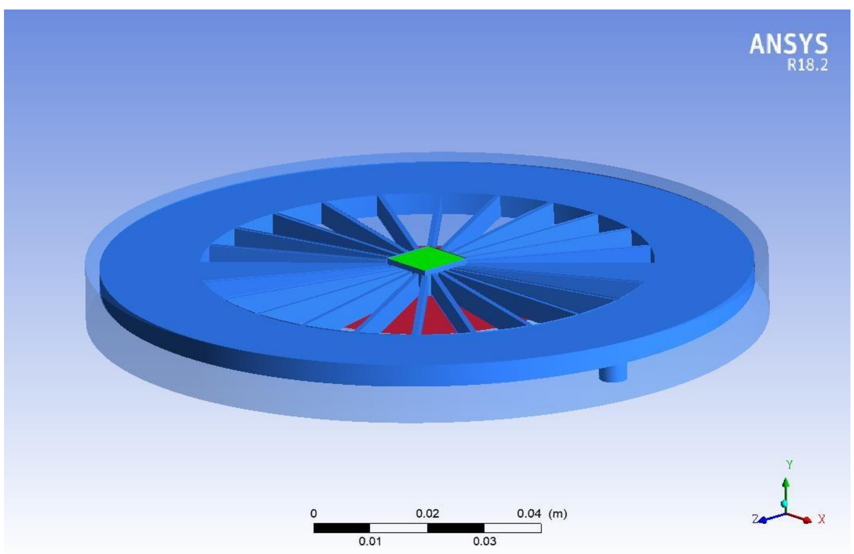

Figure 4. 3D Figure Rectangular imported in ANSYS Software.

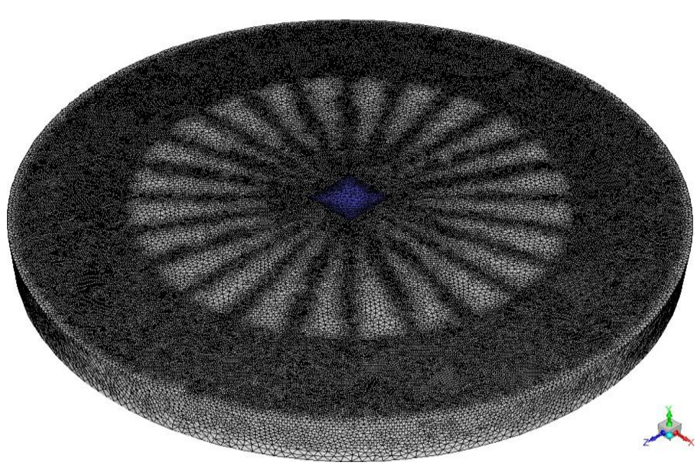

Figure 5. Solid view of Mesh

4. Result and Discussion

The effects of input parameters have been depicted on pressure through the visual effects that have been created by the ANYSY software. The effect on pressure is shown with the change in flow rate. For the fixed inlet plenum dia of 10 mm flow rate is increased from 0.25 lpm to 0.5 lpm and thento 0.75 lpm, the variation in pressure is recorded as under:

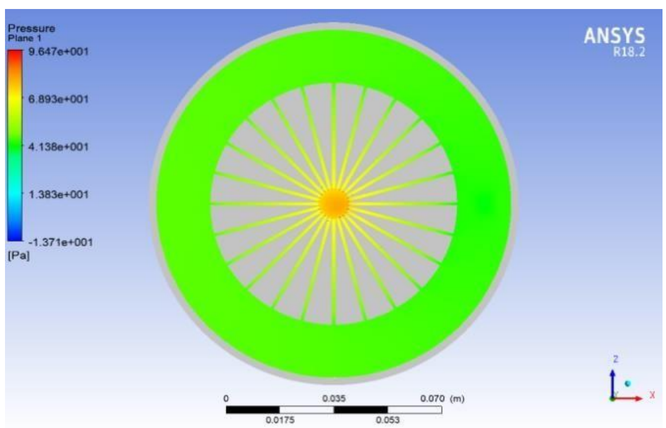

Figure 6. Pressure visualization for 0.25 lpm flow rate and 10 mm inlet plenum diameter

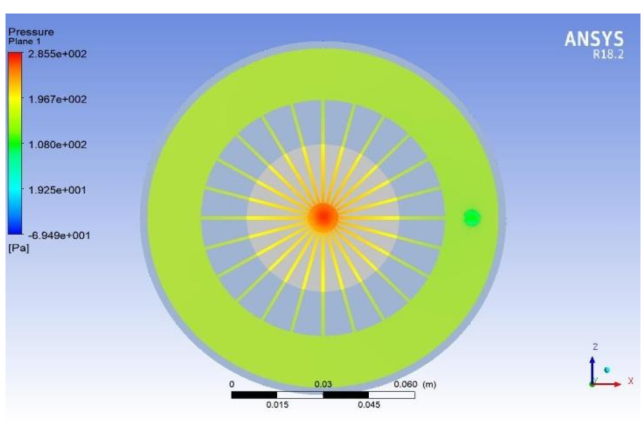

Figure 7. Pressure visualization for 0.50 lpm flow rate and 10 mm inlet plenum diameter

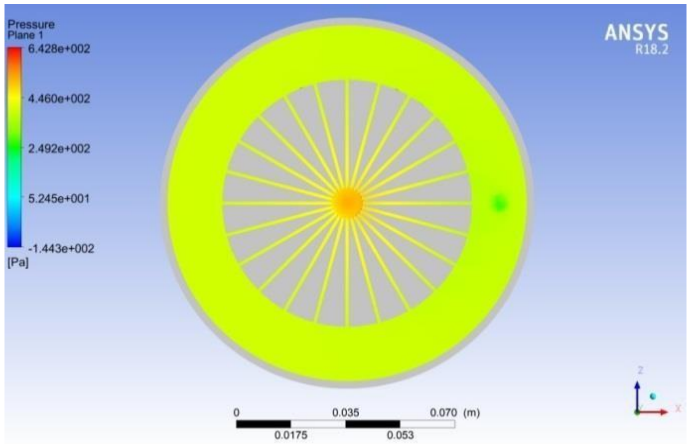

Figure 8. Pressure visualization for 0.75 lpm flow rate and 10 mm inlet plenum diameter

5. Conclusions

Subsequent are the conclusions comes from the experimental research on pressure, temperature and velocity using response surface methodology technique.

- With the increase in inlet plenum diameter the pressure decreases
- Temperature increases with the increase in the plenum diameter
- The velocity is first increases by increasing diameter from 10 mm to 20 mm and after that the velocity is decreases with increase in diameter from 20 to 30 mm diameter.
- With the increase in the size of the inlet plenum Diameter the Nusselt number increases and is maximum for inlet diameter of 30mm
- With the increase of aspect ratio the mean of Nusselt number is decreases.
- The pressure is increases significantly with increase in aspect ratio indicating that it is dominant factor affecting response that is pressure in radial micro channel. At large value of aspect ratio means, more volume of water per second which implies that flow rate increase and increase the pressure in radial micro channel.
- The Temperature is decreases by rise in aspect ratio indicating that it is dominant factor affecting response that is temperature in radial micro channel.
- There is little variations are observed in velocity by varying the aspect ratio.

6. Future Scope

Present work is helpful in finding the optimum solution for cooling of electronics chips by radial micro channel having the parameters of aspect ratio, diameter, and flow rate of fluid for optimize the properties of fluid i.e. pressure, velocity and temperature. The following areas are identified as future scope of work:
The variation in the material can be one area for future research.

- More CFD analysis can be done for different size of the micro channels having different aspect ratios.
- The effect of roughness on the performance of the radial micro channel heat sink is also open for the researchers.
- Different profile of the channels within remains unexplored till date.

References

[1]. C. Zapata and P. Nieuwenhuis, Exploring innovation in the automotive industry: new technologies for cleaner cars, J. Clean. Prod. 18(1) (2010) 14-20.

[2]. K.V.S. Rao, S.N. Kurbet and V.V Kuppast, A review on performance of the IC engine using alternative fuels, https://doi.org/10.1016/j.matpr.2017.11.303

[3]. R.F. Patterson and R.A. Harley, Evaluating near-roadway concentrations of diesel- related air pollution

using RLINE, Atmospheric Environment, 199 (2019) 1-184, https://doi.org/10.1016/j.atmosenv.2018.11.016
[4]. S.P. Singh and D. Singh, Bio-diesel production through the use of different sources and characterization of oils and their esters as the substitute of diesel: A review, https://doi.org/10.1016/j.rser.2009.07.017
[5]. Global Energy Statistical Yearbook 2018, Enerdata (2019), Retrieved from: https://yearbook.enerdata.net/total-energy/world-consumption-statistics.html
[6]. Global Energy Statistical Yearbook 2018, Enerdata (2019), Retrieved from: https://yearbook.enerdata.net/total-energy/india-consumption-statistics.html
[7]. T. Shah, A History of Petrol, (2018), Retrieved from http://fuelsucks.com/uncategorized/a-history-of-petrol/
[8]. Navajo, Petroleum Advantages and Disadvantages, (2015), Retrieved from https://navajocodetalkers.org/9-petroleum-advantages-and-disadvantages
[9]. H. Jaaskelainen, Early History of the Diesel Engine, (2013), Retrieved from https://www.dieselnet.com/tech/diesel_history.php
[10]. W. AddyMajewski, What is Diesel Fuel, (2016), Retrieved from: https://www.dieselnet.com/tech/fuel_diesel.php
[11]. Matthew C Keegan, Pros and cons of Diesel Fuel, (2015), Retrieved from http://knowhow.napaonline.com/the-pros-and-cons-of-diesel-fuel/
[12]. Thomas, Alternate fuels, (2017), Retrieved from https://www.fueleconomy.gov/feg/current.shtml
[13]. ADFC, Alternate Fuels and Data Centre, Ethanol, (2017), Retrieved from https://afdc.energy.gov/fuels/ethanol.html
[14]. P. Rodriguez, R. Melo, E. A. Goyos-Perez and L. Verhelst, Conversion of by- products from the vegetable oil industry into bio-diesel and its use in internal combustion engines: a review, Braz. J. Chem. Eng., (2014),http://dx.doi.org/10.1590/0104-6632.20140312s00002763
[15]. ADFC, Alternate Fuels and Data Centre, Natural Gas, (2017), Retrieved from https://afdc.energy.gov/fuels/natural_gas.html
[16]. ADFC, Alternate Fuels and Data Centre, Hydrogen, (2017), Retrieved from https://afdc.energy.gov/fuels/hydrogen.html
[17]. ADFC, Alternate Fuels and Data Centre, Electricity,(2017), Retrieved from https://afdc.energy.gov/fuels/electricity.html
[18]. ADFC, Alternate Fuels and Data Centre, (2017),Bio-diesel, Retrieved from https://afdc.energy.gov/fuels/bio-diesel.html
[19]. I.S.A. Manaf, N.H. Embong, S.N.M Khazaai, M.H.A.B. Rahim, M.M. Yusoff, K. Teonglee and G. Pragasmaniam, A review for key challenges of the development of bio-diesel industry, Energy Convers. Mgt. 185 (2019) 508-517.