CHAPTER 2

Literature Review

Turbulent wall jet is widely used in different engineering applications (discussed in Chapter 1), wherever heat or mass transfer is required. Because of its vast applicability and interesting characteristics, the turbulent wall jet is popular among researchers to explore. The turbulent wall jet was first experimentally explored by Forthmann [26], where he discussed that the velocity profile shows self similar behaviour when the velocity is divided by local maximum velocity and the positional coordinate normal to wall is divided by the distance from the wall in the outer shear layer where velocity becomes half of the local maximum velocity. He also explained the shear stress profile for the wall jet. Further, Glauert [32] analytically studied the wall jet. He divided the turbulent wall jet into two different layers for the analysis: the boundary layer which is near the wall and the free layer which is closer to the surrounding. In the early '80s, Launder and Rodi [60, 61] summarized the numerical and experimental studies on turbulent wall jets. They have noted that the flow field of wall jet is a combination of two shear layers, the outer shear layer and the inner shear layer. The outer shear layer of the wall jet behaves like a free jet and the inner shear layer behaves like the conventional wall boundary layer. They mentioned that for the plane wall jet, there is approximately 30% less rate of linear growth of the outer shear layer than the free jet. They divided the development of turbulent jet into three zones. The first zone is the potential core zone where the velocity and temperature remain the same and equal to the inlet condition. This region is generally formed between the jet exit to $X \leq 4-5$. The second zone starts as soon as the potential core is exhausted; the intermixing of fluid is very high in this zone and the decay rate of velocity and temperature is also high in this zone. This zone spreads between $X = 4-5$ to $X = 20$. With the passing of time, for developing a highly efficient, closely packed and reliable cooling or heating system, the intensification of heat transfer has become necessary. In this regard, the turbulent wall jet gained popularity among many researchers and they started working on increasing the heat dissipation rate of the target plate using the turbulent wall jet. A plethora of research has been carried out to enhance the rate of heat transfer by using active and passive techniques for turbulent wall jets. The active method requires external power to enhance the heat transfer by increasing the Reynolds number [18, 33, 38, 39, 105, 106], surface vibration [69], using external flow [10, 11, 31, 49, 51, 66, 88] etc. While in the passive method, no external power is required for heat enhancement, which makes it preferable over the active method. This is achieved through changing inlet parameters [4, 33, 51, 62, 85, 87], use of additives [28, 46, 79], changing surface geometry [19, 22, 35, 47, 89, 91, 92], wall inclination [39, 59, 74, 93, 101] and surface roughness [5, 13, 83, 98] etc. In this chapter, the literature is summarised for the influence of different parametric or structural changes on fluid flow and thermal characteristics.

2.1 Literature on active techniques

2.1.1 Reynolds number

Godi et al. [33] have experimentally investigated the turbulent wall jet with a square and a circular nozzles and the Reynolds number is varied from 5000 to 20000. The main aim of their invistigation is to find out the influence of hydraulic diameter, shape of jet and Reynolds number on the different thermal and fluid flow parameters. They have used hotwire anemometer for the mean and fluctuating velocity measurement and FLIR camera for the temperature measurements. They have observed that for a given shape of the nozzle, the local Nusselt number increases with the increasing Reynolds number. The influence of Reynolds number on heat transfer is more prominent in the near flow field. And for a fixed Reynolds number, the circular jet gives relatively higher heat transfer in comparison to the square jet. It is concluded that the circular jet has a slower velocity decay rate and a higher axial momentum. Hnaien et al. [39] have studied the influence of Reynolds number on the heat transfer for isothermal and isoflux inclined wall. They have performed their numerical study using RANS standard $k-\omega$ model on Fluent software. The Reynolds number is varied from 15000 to 40000 and the inclination angle is varied from 0 to 25 degrees. They have concluded from their results that for $15000 \leq Re \leq 40000$, the average Nusselt number decreases as the inclination angle increases. For the inclination angles between 0 and 25, the average Nusselt number increases as the Reynolds number increases. The influence of Reynolds number on local and average Nusselt numbers has also been studied by Hnaien et al. [38] for an offset jet, where Reynolds number is varied from 10000 to 40000. The influence of Reynolds number on heat transfer rate has also been studied numerically using low Reynolds number model by Kechiche et al. [51] for the turbulent jet with isoflux wall. From the results, they have concluded that as the Reynolds number reduces, there is a decrease in convective heat transfer between the wall and the jet. They have also studied the influence of different inlet profiles, i.e. triangular, parabolic and uniform on the flow and heat transfer characteristics. They found that the local maximum streamwise velocity and heat transfer rate are higher for the triangular profile followed by the parabolic and uniform profile in the near field. In all the above studies [33, 38, 39, 51], one thing is common, i.e. the heat transfer rate increases by increasing the Reynolds number; but for that, extra power is required. Some of the important literature is also mentioned, where the Reynolds number effect of turbulent jet on different fluid flow characteristics has been studied. The influence of Reynolds number (10000-30000) has been studied experimentally by Villafruela et al. [101]. For measuring the mean and fluctuating velocities, they have used hotwire anemometer. They observed that the spread rate and the decay rate of maximum streamwise velocity decrease as the Reynolds number increases. Akfirat [6] has experimentally studied the plane wall jet over an isothermal wall for the Reynolds numbers between 190 to 16,620. The local heat transfer has been calculated for different Reynolds numbers and for two slot widths of 1.27 mm and 2.54 mm. The effect of Reynolds number has been numerically studied by Bogey and Bailly [18] for the circular jet using large eddy simulation (LES). For the study, the Reynolds number is varied from 1,700 to 4,00,000. They have found that the velocity profile shows self similar behaviour early for the lower Reynolds numbers when flow moves forward in the streamwise direction. The decay rate of streamwise centerline velocity decreases as the Reynolds number increases. In the near flow field $X \leq 15$, the streamwise turbulent fluctuating velocity u'_{rms} increases with the increase in Reynolds number. However, in the locations $15 \leq X \leq 25$, the turbulent intensity is higher for the low Reynolds number jet. Suresh et al. [96] have conducted experiment to study the influence of Reynolds number on the transitional characteristics of a rectangular wall jet. The Reynolds number is varied from 250 to 6250 and a hot wire anemometer has been used for the measurement of mean and fluctuating velocities. For this range of Reynolds number, the flow characteristics are highly dependent on the Reynolds number. However, in the case of higher Reynolds number, there is no influence of Reynolds number in the far flow field. Xu et al. [105, 106] have experimentally explored the influence of Reynolds number on the turbulent jet for 5 different Reynolds number between 8000 to 50000. Their study shows that the fluid flow characteristics are dependent on the Reynolds number till Re=30000; beyond this, the influence of Reynolds number is almost negligible.

2.1.2 External flow

In the external flow condition, the surrounding fluid does not remain stationary and it flows with a certain velocity. When the external flow is in the direction of the main jet then it is called co-flow condition and when the direction of external flow and jet flow is opposite, it is called counter flow. Naqavi et at. [66] have performed numerical simulation using large eddy simulation for a wall jet with the external flow. The jet velocity ratio $M = U_0/U_\infty$ is varied between 0.3 to 2.3, where U_0 is the velocity of jet. It is reported that for the velocity ratio $M < 1$, the heat transfer rate increases with the increasing jet velocity ratio and maximum heat transfer is achieved at $M = 1.05$ and the heat transfer rate decays slowly on further increasing the jet velocity ratio. The influence of co-flow directed at different angles with respect to the axis of the jet has been numerically investigated by Ayech et al. [11] using low Reynolds number $k-\varepsilon$ model. The study is done by varying the co-flow inclination angle from 0ř to 60ř and the velocity ratio $r = U_{co}/U_0$ is varied between 0.02 to 0.4. They observed that the jet decays at a faster rate when combined with the co-flow, and by increasing the deviation angle of co-flow, the jet growth rate and the turbulence mixing also increase which enhances the heat transfer rate. Gao and Ewing [30] and Goa et al. [29] have conducted an experiment to study the influence of co-flow wall jet on the flow and thermal characteristics of a turbulent offset jet with the Reynolds number equal to 42000. They have used single and cross wire hotwire probes for the velocity measurement; a pressure transducer and microphones are used to measure mean static pressure and fluctuating static pressure. The wall jet with the counter flow has been numerically examined by Sharma et al. [88]. The self similarity is noticed for the shear stress and mean velocity profiles up to the location $X = 50$ and $X = 55$, respectively. They have also concluded that the counter flow helps in increasing the turbulence diffusion of fluid. The influence of co-flow has been experimentally studied in the near flow field $X \leq 10$ by Kaffel et al. [49]. The main motive of this study was to explore the effect of co-flow for the early stage of the transition zone where intermixing of fluid is very high. For the study, the co-flow velocity ratio $r = 0, 0.16$ and 0.33 has been used. They have concluded that with the increase in co-flow velocity ratio, the entrainment of surrounding fluid decreases and the decay of streamwise mean velocity increases.

2.2 Literature on passive techniques

2.2.1 Inlet parameters

Recently, Godi et al. [34] have experimentally and numerically studied the heat transfer characteristics of a single and row jet, where the size of the jet is changed from 1.5 mm to 3 mm and the space between the two consecutive jets are kept 2D and 4D (D=hydraulic diameter). They have used 4 low Reynolds number models, viz. (i) Spalart Almaras (SA) model (ii) Realizable $k-\varepsilon$ model (iii) $k-\omega$ SST model and (iv) Reynolds Stress Model to find the best model for their numerical results. For a fixed Reynolds number, they found that the size of the jet and the spacing between the jet have negligible influence on the heat transfer rate in the near flow field region. However, for a fixed mass flow rate, the jets with 4D spacing have higher heat transfer rates. They have also investigated the different sizes and shapes of the nozzle experimentally for square and circular nozzles with the hydraulic diameters of 2.5 mm and 7.5 mm [33]. From this study, they found that the nozzle shape influences the hydraulic and thermal characteristics in the near field $X \leq 10$ only, whereas in the case of average heat transfer rate, the influence of size is more prominent than the shape. For the same hydraulic diameter, the local Nusselt number is higher for the circular jet than the square jet. The experiment has been carried out to examine the influence of tabs and chevrons at the nozzle exit by Sexton et al. [87] by using 2-D PIV facility. They mentioned a 29% enhancement of the average Nusselt number in comparison to the simply slotted jet. The use of tabs at the inlet leads to the enhancement of mixing of the jet and the ambient fluid [85, 108, 109]. The obstruction caused by the solid surface of tabs creates the pressure gradient near the inlet which gives rise to the formation of streamwise vorticity. The streamwise vorticity thus generated increases the mixing of the jet fluid with the surroundings and the jet spreads at a higher rate in comparison to the nozzle with no tabs. Le et al. [62] have experimentally and numerically studied the influence of jet axial distance from the plate on heat transfer and fluid flow characteristics for axial distances 0a, 3a and 6a. For the experimental study, they have used 1-D hotwire anemometer and for the numerical simulation they have used $k-\varepsilon$ model with standard wall function. They found that there is no effect of jet distance on centerline mean velocity as the centerline velocity profiles are similar in all the cases but, the heat transfer rate of larger jet to wall distance is higher than the smaller one. The influence of turbulent intensity level at the inlet and its cross stream distribution on heat transfer rate is numerically analysed by Adeniji et al. [4] for a plane wall jet. They carried out simulation using the standard $k-\varepsilon$ model for a two-dimensional geometry for different turbulent intensities from 0.5% to 5%. The result revealed that as turbulent intensity increases from 0.5% to

5%, the local Nusselt number increases in near field $X \leq 15$; beyond that, all the cases follow the same trend. The effect of cross stream turbulent intensity at the inlet is also limited to the near field $X \leq 15$ only. The effect on heat transfer due to the inlet turbulent intensity is more pronounced than the cross stream distribution at the inlet. The influence of inlet conditions on heat transfer rate has also been studied numerically using standard $k-\varepsilon$ model by Kechiche et al. [51] for the turbulent plane wall jet. Triangular, parabolic and uniform inlet profiles are investigated for the Reynolds numbers 14400, 16000 and 18000. They found that the local maximum streamwise velocity and heat transfer rate are higher for the triangular profile followed by the parabolic and uniform profile in the near field. The above discussion [4, 33, 51, 62, 87] reveals that modification in inlet conditions using the passive method has influence on flow and heat transfer characteristics in the near flow field only. Mclntyre et al. [65] have experimentally studied the effect of nozzle upper lip thickness, varied from 0.125a to 2a, on flow characteristics. A Dantec Dynamics Cross hotwire anemometer is used to study the mean and turbulent velocity and the Reynolds stress. They have noted that the value of X normal Reynolds stress increases as the nozzle thickness increases from 0.125a to 2a at the location of $X = 10$, but the spread rate remains unaffected. Ioannou and Laizat [44] have numerically investigated plasma control jet using Implicit Large Eddy Simulation. The results revealed the decrease in potential core and improvement in mixing properties of fluid flow. Sun and Ewing [94] have studied the influence of initial and boundary conditions on the turbulent wall jet. They used a nozzle of short length to get a top hat (uniform) velocity profile and a nozzle of longer length for a fully developed velocity profile (follows 1/7th law) at the inlet. Kumar and Kumar [55] have also investigated the effect of initial condition of turbulent wall jet by changing the nozzle length to $l/h = 10$, 50, and 90. They found that for the shorter nozzle l/h=10, the decay of maximum velocity is faster in the near flow field.

2.2.2 Additives

The additives have also been used to increase the removal of heat from the target plate [28, 46, 79, 100]. The flow and heat transfer characteristics of a turbulent jet using molten salt have been explored experimentally by Gao et al. [28]. They found that the molten salt jet shows heat transfer characteristics close to the water jet and for the same Reynolds number, the molten salt jet has a higher heat transfer rate. For the increasing Reynolds number, the increase in heat transfer rate has been noticed for the molten salt. The influence of different nanofluids (Ag, Cu, CuO, Al_2O_3 and TiO_2) on heat transfer has been numerically studied by Turkyilmazoglu [100] for the plane wall jet. He observed that the skin friction is highest for the Ag in comparison to Al_2O_3 nanofluid. The Ag nanofluid has the highest heat dissipation rate and TiO_2has the lowest. However, in spite of having very high heat dissipation rate, the applicability of nanofluid in the turbulent jet is limited as it can not be used in an open environment.

2.2.3 Surface roughness

The fluid flow characteristics of a turbulent wall jet on a rough surface have been extensively explored by Rostamy et al. [82, 83]. They used Laser Doppler Anemometry to experimentally investigate transitionally rough surface for a Reynolds number of 7500. They concluded that the effect of transitionally rough surface is limited to the inner shear layer only. The magnitude of both the normal Reynolds stresses (u'u' and v'v') decreases in the inner shear layer when inner scaling is utilized in comparison to the smooth surface. The study on the fully rough surface has been conducted experimentally by Tang et al. [98]. They used Particle Image Velocimetry (PIV) to study a fully rough surface for a Reynolds number of 13400. They noted a delay in the formation of developed zone in comparison to the smooth surface from $X = 40$ to $X = 70$. They also noted that the inner shear layer thickness increases in case of rough surface due to which the position of maximum streamwise velocity shifts away from the wall. Tachie et al. [97] have investigated the effect of roughness on the flow characteristics of a turbulent wall jet. They have concluded that the skin friction coefficient increases in the case of rough surface, although, the spread of jet remains unaffected by the roughness of the wall. Afzal and Seena [5] have done analysis of turbulent wall jet on the transitional rough, fully rough and smooth surfaces. Wu and Rajaratnam [102] have studied the influence of surface roughness on the mixing and spreading of turbulent jet with Reynolds numbers 5,160 and 8,110. They have used surface flower and strip patterns for the surface roughness. They have found that there is a linear growth in the skin friction coefficient with respect to the Roughness of the wall. The turbulent wall jet flowing over the smooth and rough surfaces has been explored experimentally by Maslov et al. [64] for high Reynolds number ranging between 25,000-125,000. They have concluded that the decay of maximum streamwise velocity increases by 20% and the inner shear layer thickness increases by 2.5 times with respect to the plane wall jet.

2.2.4 Wall inclination

The heat transfer and fluid flow behavior of turbulent offset oblique wall jet have been explored experimentally by Song et al. [93]. They reported that as the oblique angle increases from 0^0 to 40^0, the difference in entrainment of fluid from the inner and outer free shear layers reduces which subsequently leads to the reduction in Nusselt number. Hnaien et al. [39] have numerically investigated the influence of inclination angle using RANS model. The inclination angle is varied from 0 to 25 degrees and the Reynolds number is varied from 15000 to 40000. They have concluded from their results that for $15000 \leq Re \leq 40000$, the average Nusselt number decreases as the inclination angle increases. Pramanik and Das [74] have investigated the fluid flow characteristics of an inclined wall jet numerically using $k-\varepsilon$ model with standard wall function. Lai and Lu [59] have examined the effect of wall inclination on the mean and turbulent flow parameters of turbulent wall jet with the help of single and cross hotwire anemometers. The inclination of wall "β" is changed from 0^0 to 90^0 with respect to the jet axis. Their results show that the decay of centerline velocity increases and the spreading of jet becomes faster as the inclination angle increases till $\beta = 45^0$. Further, Pramanik and Das [75] have numerically studied the turbulent wall jet flowing over a multiple inclined wall by the help of standard $k-\varepsilon$ model, where the wall is made of 4 plane wall with inclination angle 5ř, 10ř, 15řand20ř consecutively.. The rate of decay of streamwise maximum velocity is higher in the case of multiple inclined wall than the plane wall jet. Pramanik and Das [73] have also numerically investigated the inclination effect of the target plate for an offset turbulent jet. Villafruela et al. [101] examined the turbulent wall jet for different inclination angles (0^0, 10^0, $20^0$and 30^0) using hot-wire anemometer. They observed that the decay rate of maximum streamwise velocity and the spread rate increase with the increasing inclination angle.

.

2.2.5 Surface geometry

Among all the parameters, the study of the modification in surface geometry has received less attention from researchers. Few studies available in the literature are mentioned here, where the flat plate is replaced by the wall with different shapes (mostly concave and convex) and the flow and thermal characteristics have been studied. The experimental investigation of turbulent jet on a plane wall and a curved wall has been done by Catalano et al. [19] in the presence of moving stream. They found that the velocity potential core gets exhausted more rapidly in the case of curved wall. The fluid flow and thermal characteristics of a turbulent jet on the heated plane and curved wall (convex wall) have been studied by Dakos et al. [22]. The experiment has been conducted for the Reynolds number of 30,000 along with the co-flow. The temperature and velocity measurements have been done by a chromel-alumel thermocouple inbuilt within the miniature hotwire anemometer. They have noted that in the case of curved surface, the skin friction coefficient reduces and the Stanton number rises in comparison to the plane wall jet. They have also observed that in the case of curved wall, the position of zero shear stress and turbulent heat flux is shifted closer to the wall and away from the position of maximum streamwise velocity. Finally, they have made the conclusion that in the case of curved wall, the turbulent intensity, turbulent heat flux, mixing of the fluid and Reynolds stresses are relatively higher than that in the plane wall jet. The experimental investigation of heated jet on plane as well as curved surfaces (concave wall) has been done by Okamoto and Gibson [72]. They observed that the spread of jet in the case of concave wall is about 80% of the spread of plane wall jet. The streamwise heat flux is slightly less than the plane wall jet. However, the cross-stream heat flux is higher in the concave wall jet. The Reynolds stresses are relatively higher for the concave wall in comparison to the plane wall, but it remained less than the convex surface. They have also mentioned that the position of zero shear stress shifts closer to the position of maximum streamwise velocity in comparison to the convex surface. The turbulent wall jet on concave surface has also been studied by Rodman et al. [81]. They have studied the influence of curvature of concave surface ($h/R = 0$ (plane wall), $h/R = 0.0031$ and $h/R = 0.029$) on different fluid flow characteristics. They have noted that in the case of small curvature $h/R = 0.0031$, the Reynolds stress remains almost similar to the plane wall jet in the flow field $X < 100$; beyond this, as the fluid moves forward the Reynolds stress increases gradually. At the location $X = 180$, an increase of 21% is noticed for the curved surface. The spread of jet in the wall normal direction and the decay rate of jet, both increase with an increase in the curvature of concave wall jet. Robert [80] have numerically investigated the turbulent wall jet on curved wall in the still surrounding. They have found that the curvature of wall plays a significant role in changing the Reynolds stress and gives rise to an adverse pressure gradient for the convex surface, which leads to separation of flow. Further, Fujisawa and Kobayashi [27] have studied the mean and turbulent flow characteristics of turbulent jet on a strong convex surface. They have concluded that the magnitude of Reynolds stresses increases with the rise in the curvature of the curved surface. The turbulent fluctuation normal to the surface has been increased due to the strong centrifugal effect.

2.2.6 Literature summary

Literature related to different parametric changes in turbulent wall jet and their impact on the heat transfer rate and other flow and thermal characteristics has been summarized in this chapter. From the literature, it is clear that by increasing the Reynolds number, the heat transfer increases and the presence of co-flow enhances the intermixing of fluid which works in favour of heat transfer rate. Although, in both the cases, an additional power supply is required. The heat transfer rate has been enhanced by changing the nozzle size, inlet velocity profile, by using tabs and chevrons at the inlet. But the modification in inlet condition using the passive method has an influence on flow and heat transfer characteristics in the near flow field only. Use of molten salt and nanoparticles also increase the heat dissipation rate but it can not be used for open surfaces. It can be concluded from the above discussions that several techniques have been employed to study the variation in heat transfer and fluid flow characteristics of a turbulent wall jet on a plane wall. However, the influence of wall pattern on heat transfer and fluid flow has been explored by very few researchers [89, 91, 92]. Shivankar et al. [89] have studied the effect of jet position for the offset jet using $k-\omega$ SST model on a wavy wall. For wavy wall, the numerical analysis is conducted by Singh et al. [91, 92] for a Reynolds number of 15000. They used the high Reynolds number standard $k-\varepsilon$ model which is not capable of capturing phenomena in viscous sublayer and is usually less accurate in the region of flow separation. In the present study, fluid and thermal characteristics which are important to understand the behaviour of the jet near the wavy wall, like $u^+ - y^+$, $T^+ - y^+$, skin friction coefficient f_x, re-circulation zone and thermal hydraulic performance (THP), are discussed which are not possible to do with standard $k-\varepsilon$ model. Moreover, in the present study, the wall has also been modified to a partial wavy wall and a partially decaying wavy wall to improve the heat transfer rate and THP.

The aim of the present work is an attempt to have a better understanding of fluid flow and heat transfer behavior of a turbulent 2D wall jet flowing over a sinusoidal wavy wall, and come up with the best pattern which gives higher heat transfer and thermal hydraulic performance (THP). The concept behind using the sinusoidal surface is to increase the surface area smoothly. One can also opt for other profiles like: triangular, wedge shaped etc. But, it is believed that more abrupt profile will lead to early separation resulting in a poor performance. The sinusoidal waveform is the most common wavy pattern we come across in daily life. Moreover, the sinusoidal wavy pattern is used in many engineering appliances to improve the heat transfer rate [8, 21].