CHAPTER 1

Introduction and Literature Review

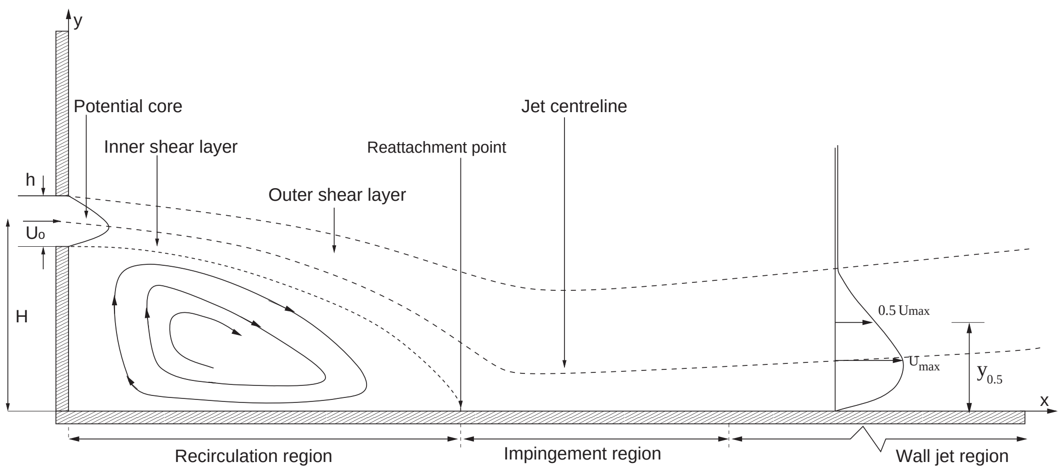
The turbulent flow encountered in practical applications are mostly wall-bounded type flow. The presence of a wall makes the flow more complicated. The formation of very thin viscous sublayer inside the boundary layer in the near-wall region is one of the distinguished characteristics of wall-bounded turbulent flows. The gradient of variables are sharp very close to the wall. Thus, very fine grids are required to capture the sharp gradient near the wall. In addition, momentum and heat transport in this layer is mainly due to the viscous diffusion. These factors make the treatment of viscous sublayer numerically challenging. Within the framework of RANS (Reynolds averaged Navier-Stokes) equations, high-Reynolds number (HRN) and low-Reynolds number (LRN) modeling employ different approaches to solve wall-bounded turbulent flow. The first near-wall grid point is placed outside the viscous sublayer in case of high-Reynolds number modeling and the wall functions are required to bridge the viscous sublayer. The low-Reynolds number turbulence models do not require the wall functions; instead, the entire boundary layer is resolved with very fine grids. However, the conventional wall functions which are widely used with RANS based high-Reynolds number turbulence models in forced convection boundary layer flows are not applicable in case of natural and mixed convection boundary layer flows. The equilibrium and other assumptions which form the basis of wall functions do not hold in case of natural and mixed convection boundary layers (Choi et al. (2004), Kenjeres et al. (2005)). Thus, low-Reynolds number turbulence models are more suitable as compared to high-Reynolds number model for natural and mixed convection type boundary layer flows.

The present work addresses the flow and heat transfer characteristics of wall-bounded jets (forced convection flow), study of effects of freestream motion on flow and heat transfer characteristics of turbulent offset jet (forced convection flow), buoyancy driven flow in a tall cavity (natural convection flow) and the buoyancy-opposed wall jet flow in a rectangular channel (mixed convection flow). Thus, force convection, natural convection and mixed convection type wall-bounded turbulent flows have been considered for study. The RANS based high-Reynolds number and low-Reynolds number turbulence models are considered for closure. The standard $k-\epsilon$ model, shear stress transport (SST) model (Menter (2009)), Wilcox $k-\omega$ model (Wilcox (2006)) and low-Reynolds number $k-\epsilon$ models proposed by Launder and Sharma (1974) (LS), and Yang and Shih (1993) (YS) are considered for numerical simulation.

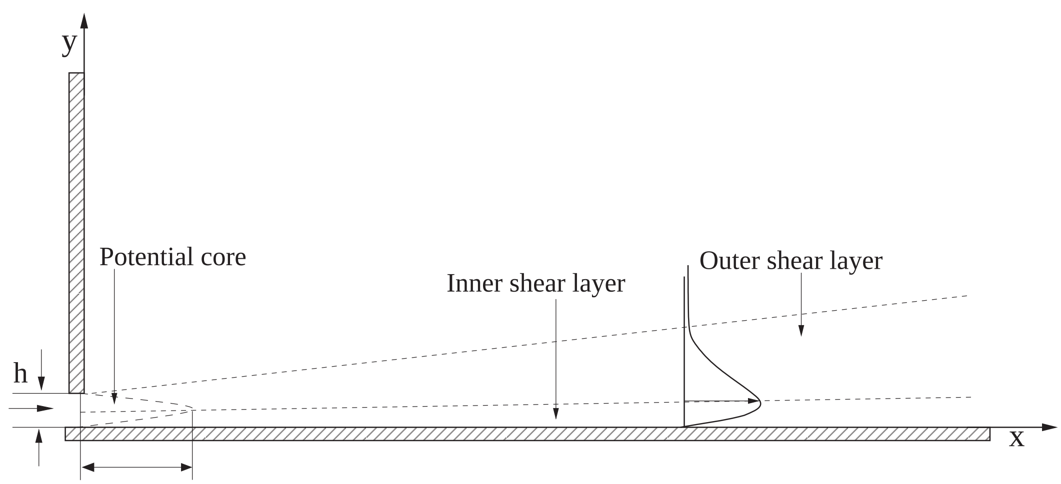
1.1 Introduction to jet

A jet is produced when a fluid is injected with the addition of some momentum from a nozzle into the surrounding fluid. The pressure drop across the nozzle results in the momentum gain of the fluid. The surrounding fluid may be moving or stagnant depending upon the application. An offset jet is produced when a jet is coming out, parallel to the axis of the nozzle but offset by a certain height from the impingement surface. Figure 1.1a shows the schematic diagram of a generic offset jet. It consists of an inner boundary layer and the outer free shear flow. The flow just downstream of jet inlet consists of a potential core region in which flow velocity remains unaffected from the viscous effects and is equal to the jet inlet velocity. The flow field of an offset jet can be classified into three regions: the recirculation region, the impingement region, and the wall jet region (as shown in Fig. 1.1a). The entrainment of the surrounding fluid above and below an offset jet is unequal due to the presence of the impingement wall. As a result, a low pressure (sub-atmospheric) region forms between the jet and the impingement wall. The jet deflects towards the wall and finally attaches at the reattachment point. This is known as the Coanda effect (Tritton (1977)). The low pressure region is known as the recirculation region. The reattachment point is a location where the wall shear stress changes its direction $(\tau_w = 0)$. The recirculation region starts from the nozzle exit and extends up to the reattachment point. The flow field of an offset jet attains the characteristics of a wall jet flow beyond the impingement region for any offset ratio (Rajaratnam and Subramanyam (1968)). The offset ratio (OR) is defined as the ratio of the jet center line height (H) to the jet width (h).

Wall jet is a special case of an offset jet when offset distance $(H-0.5h)$ becomes zero. The schematic diagram of a wall jet is shown in Fig. 1.1b. The flow field



(a) An offset jet



(b) A wall jet

Figure 1.1: Schematic diagram of an offset jet and a wall jet

of a plane turbulent wall jet consists of an inner shear layer similar to the wall boundary layer and an outer shear layer. The flow velocity increases from zero at the wall to a local maximum streamwise velocity inside the inner shear layer. Inside the outer shear layer, the flow velocity decreases from the local maximum streamwise velocity to a zero velocity in case of a quiescent ambient. The outer layer is known as the free mixing region (Rajaratnam (1976)).

1.2 Applications of jets

Turbulent jets play an important role in many engineering devices, some of which include: flow deflection devices, boundary layer separation control by mixing wall jet to provide additional momentum, thrust augmentation in aircraft during vertical take-off. The wall jet is most widely used in automobiles defrosters where wall jets are used to augment heat and mass transfer from the surface. Film-cooling technology is the other important area where turbine blades in gas turbine, and boiler combustion chamber are protected from hot gases. Some more examples which may be cited are: cooling of electronic components in high heat load applications, fuel injection systems, heating and air conditioning applications, heat exchangers, etc. The turbulent jet flow in many situation is accompanied by the presence of a co-flow e.g. disposal of effluent jet coming from waste water outlets in river and marine, wakes of aircraft, emission of pollutant from chimneys, etc.

1.3 Literature review

1.3.1 Literature related to turbulent offset jet and wall jet flows in the quiescent medium

The first detailed investigation of turbulent plane offset jet is reported by Bourque and Neumann (1960) in their pioneering work in the year 1960. The study is concentrated on the measurement of static pressure and velocity field. They have reported the effects of Reynolds number and offset ratio on reattachment length, and wall static pressure in the recirculation region. They have also found by dimensional analysis that for large Reynolds numbers, the impingement distance becomes independent of Reynolds number. The flow field and thermal characteristics of an offset jet have been studied by several authors. Sawyer (1960) has experimentally studied the turbulent offset jet flow to determine the average pressure and length of recirculation region. They have approximated the curved portion of a plane offset jet as a plane free jet in their theoretical model and the jet centerline is considered as a circular arc. Sawyer (1963) later on modified the previous study (Sawyer (1960)) by taking into account the different rates of entrainment by the two edges of the curved jet.

Pelfrey and Liburdy (1986) experimentally studied the mean flow and turbulence characteristics of a turbulent offset jet. The measurement was done using laser Doppler anemometry (LDA) for offset ratio and Reynolds number 7 and 15000, respectively. The offset ratio (OR) is defined as the ratio of the jet center line height (H) to the jet width (h) and the Reynolds number (Re) is based on the jet inlet mean velocity (U_0) and the jet width (h). They measured the mean velocity profile at different axial locations, ratio of curvature to shear strain rate, and the entrainment parameter in the recirculation and impingement regions. They mentioned that the flow cannot be modeled as thin shear flow as the ratio of curvature to shear strain rate, and ratio of jet width to radius of curvature were considerably higher in the recirculation region. In conclusion, the flow is subjected to additional large strain rate due to jet curvature in the recirculation and impingement regions that makes it a challenging test case for testing of turbulence models. Holland and Liburdy (1990) later on experimentally investigated the heat transfer characteristics of heated offset jet and wall jet flows for a flow geometry similar to that of Pelfrey and Liburdy (1986). The objective was to provide detailed experimental investigation of heat transfer characteristics of turbulent offset jet as literature dealing with heat transfer characteristics of turbulent offset jet was very scarce. The offset ratios considered were 3, 7 and 11. The temperature measurements were done with a very fine thermocouple wire probe (0.0254 mm chromel and alumel wire) in the flow field. They measured the temperature profile at different axial locations, decay of the local maximum axial temperature and temperature variation along the impingement wall in the recirculation, impingement and developing wall jet regions. They observed that temperature within the recirculation region was approximately uniform for the range of offset ratios investigated. The other important outcome from the experiment was that the flow experiences greater strain rate with higher offset ratio and it increases with increase in offset ratio.

Kim et al. (1996) have experimentally investigated the flow and heat transfer characteristics of turbulent offset jet for offset ratios in the range $0-20$ and Reynolds number in the range $6500-39000$. They have carried out measurements of the mean velocity and turbulent intensity, and wall temperature using split film probe and thermochronic liquid crystal, respectively. They have found that the point of maximum Nusselt number coincides with the time averaged reattachment point. They have observed the presence of a secondary vortex which causes better mixing and increases the Nusselt number near the corner. Nasr and Lai (1997) have experimentally studied the flow and turbulence characteristics of two parallel plane jets and an offset jet under identical flow conditions. They have performed comparative analysis of two parallel plane jets with separation ratio 2.125 and the offset jet with offset ratio 2.125 using two component LDA. They have observed that the recirculation zone is smaller for the two parallel jets than for the offset jet. They have concluded that the presence of solid wall in case of the offset jet have significant retarding and suppression effects on flow and turbulence fields in the recirculation and impingement regions.

The study of the mean flow field and turbulence characteristics for a small offset ratio 2.125 and Reynolds number of 11000 has been done by Nasr and Lai (1998). The experimental study has been performed using LDA, whereas for numerical simulation, three different turbulence models (the standard k-ϵ, RNG and Reynolds stress models) have been considered. The numerical results have been compared with the experimental results to predict the capability of different turbulence models. The effects of various discretization schemes (power law scheme, second order upwind scheme, and QUICK scheme) on the reattachment length have also been investigated. All turbulence models have predicted the reattachment length better when the power law scheme is used. They have finally concluded that the standard $k - \epsilon$ model is more appropriate for the prediction of turbulent offset jets with small offset ratios among the three models considered.

The wall jet is a special case of an offset jet when offset distance becomes zero as discussed earlier. The first experimental study of a turbulent plane wall jet was carried out by Forthmann (1936). The self-preserving nature of wall jet and self-similarity of axial velocity components were reported. The self-preserving wall jet maintains a constant ratio of maximum velocity to freestream velocity along the flow direction. The wall jet flow in a quiescent surrounding is self-preserving as the ratio of maximum velocity to freestream velocity is constant and infinity along the flow direction. However, the wall jet flow ceases to be self-preserving in the presence of a uniform freestream due to continuous decay of local maximum velocity along the flow direction. The similarity problem of radial and plane wall jets was theoretically studied by Glauert (1956) for laminar and turbulent flows. The similarity was explicitly obtained for laminar flow. However, complete similarity was found to be unattainable for turbulent flow. The detailed experimental study of mean velocity distribution of turbulent plane wall jet was reported by Schwarz and Cosart (1961) with the help of hot wire anemometer. They had shown theoretically that the local maximum axial velocity decays as x^a, where x is axial coordinate. The exponent a had been empirically obtained equal to -0.555. The literature on wall jets are very large due to many practical applications. Launder and Rodi (1981), (1983) had reviewed the experimental work related to turbulent wall jet flow up to the year 1980.

The experimental work conducted by AbdulNour et al. (2000) provides the temperature profile in the thermal sublayer which is very scarce in the literature. The experimental data for temperature profile in the thermal sublayer are very crucial to judge the suitability of different low-Reynolds number turbulence models in the near-wall region. AbdulNour et al. (2000) experimentally investigated the heat transfer characteristics of a wall jet ($Re = 7700$) for isothermal and constant heat flux boundary conditions. The measurements were done using micro-thermocouples and Infrared (IR) imaging. The temperature measurement in case of the isothermal boundary condition was done using micro-thermocouple, whereas in case of the constant heat flux boundary condition both micro-thermocouple and IR imaging were used. They measured the temperature profile in the thermal boundary layer including the thermal sublayer at different axial locations. Furthermore, the axial variation of the local heat transfer coefficient for isothermal and constant heat flux boundary conditions was also measured. They observed that the axial variation of the local heat transfer coefficient was insensitive to the thermal boundary conditions for axial locations $X \geqslant 5$. They suggested IR imaging for more accurate measurement of the local convective heat transfer coefficient in case of the constant heat flux boundary condition.

Kechiche et al. (2004) numerically predicted the flow and thermal characteristics of the turbulent wall jet flow for Reynolds number in the range $7300 - 22500$. Different thermal boundary conditions considered were isothermal jet or jet submitted to the impingement wall with different boundary conditions (isothermal or constant heat flux). Low-Reynolds number $k - \epsilon$ models proposed by Chien (1982), Nagano and Hishida (1987) and Herrero et al. (1991) were considered for numerical simulations. They compared the results of numerical simulation with the experimental results available in the literature. They concluded that for the case of non-isothermal wall jet, Herrero et al. (1991) model was more effective for predicting the thermal characteristics. Seyedein et al. (1994) numerically investigated the flow and thermal characteristics of a confined impingement turbulent slot jet. The standard $k - \epsilon$ model and low-Reynolds number $k - \epsilon$ models proposed by Lam and Bremhorst (1981) and Launder and Sharma (1974) were considered for numerical simulation. They considered Reynolds number in the range $5000 - 20000$ and nozzle to jet spacing in the range $2.5 - 7.5$. The Nusselt number obtained from different models were compared with the experimental results. Based on the comparison, it was found that the low-Reynolds number $k - \epsilon$ models performed better as compared to the standard $k - \epsilon$ model. Launder and Sharma (1974) model showed better agreement with the experimental results among the models considered.

El-Gabry and Kaminski (2006) numerically and experimentally investigated the three-dimensional jet impingement with cross flow for Reynolds number in the range $14733 - 34878$. The jet angle was varied between 30^0, 60^0 and 90^0 as measured from the impingement surface. The objective was to examine the performance of Yang and Shih (YS) (1993) low-Reynolds number $k - \epsilon$ model and the standard $k - \epsilon$ model for the prediction of heat transfer characteristics of jet in cross flow. They compared the numerical results obtained from YS model and the standard $k - \epsilon$ model using Fluent with their experimental results. They observed that the standard $k - \epsilon$ model performed better in some cases while the YS model preformed better in others. Finally, they concluded that there is no single turbulence model that performs better in all cases. The errors in predicting the average Nusselt number were $2 - 30\,\%$ for the YS model and $0 - 60\,\%$ for the standard $k - \epsilon$ model as compared to the experimental results. The jet impingement on a surface having a constant heat flux over a limited area was numerically investigated by Shuja et al. (1999) for $Re = 23000$ and 70000. They considered the low-Reynolds number $k - \epsilon$ model proposed by Lam and Bremhorst (1981), standard $k - \epsilon$ model and two Reynolds stress models for turbulence closure. They observed that temperature profiles predicted by the low-Reynolds number $k - \epsilon$ model and Reynolds stress model were better as compared to the standard $k - \epsilon$ model.

1.3.2 Literature related to turbulent offset jet and wall jet flows in an external stream

Many authors have studied the effects of co-flow on dynamics and mixing of turbulent free and wall jet flows experimentally or computationally. The free jet in the presence of a co-flow stream has been extensively studied in the literature. Some of the important work are: Maczynski (1962), Bradbury and Riley (1967), Antonia and Bilger (1973), Gaskin and Wood (2001), Antoine et al. (2001), Habli et al. (2008). Nickels and Perry (1996) experimentally and theoretically studied the turbulent axisymmetric co-flowing jet. The experimental setup comprising wind tunnel was designed to issue the jet from a 25.4 mm nozzle and a maximum velocity of 3.5 m/s. The mean flow measurements were carried out using both pitot-static tubes and dynamically calibrated hot wires. They had proposed a simple, crude model which gives the correct qualitative variation of radial profiles of Reynolds stresses, mean velocity profiles as compared to the experimental results. Recently, Habli et al. (2014) have carried out the computational study of a turbulent plane jet flow in a co-flow environment. They have considered co-flow velocity ratio up to 0.1 and standard $k - \epsilon$ model for closure. They have compared some of their computational results with the experimental results of Deo et al. (2007). They have revealed that velocity decay rate is lower for higher values of co-flow velocity ratio. They have also reported that parameters can reach an asymptotic curve at different co-flow velocity ratios when using a momentum length scale.

The experimental study of flow and turbulence characteristics of plane wall jet in a moving stream has been carried out by Kurka and Eskinazi (1964). They have considered co-flow velocity ratio up to 0.485 and Reynolds number up to 26270. The turbulence measurements have been taken with a multiple channel constant current hot-wire anemometer while the wall shear has been measured with a flattened Preston tube. They have divided the flow at U_{max} locations for mean measurements and at the locations of $\overline{u'v'} = 0$ for statistical quantities. They have found a linear relation between shear in the free-mixing region and the maximum excess velocity which can be utilized to calculate the turbulent shear from mean measurements. They have also found that u_τ is proportional to U_{max}. Irwin (1973) has carried out the experimental study of turbulent plane wall jet in an adverse pressure gradient in the presence of a freestream. He has done the measurement with a linearized hot-wire anemometer for Reynolds number of 28000. He has found that production of turbulent kinetic energy is not negative near to the velocity maximum however point of zero shear stress is always closer to the wall than the point of maximum velocity. He has reported that the mean velocity profile is logarithmic with constants that are similar to the conventional values in wall-bounded boundary layer flows. The theoretical study of plane wall jet in a co-flow stream has been carried out by Campbell (1975). He has modeled the wall jet flow using an integral model which includes turbulent shear stress, entrainment and heat transfer. He has applied the velocity profile suggested by Escudier and Nicoll (1966) and solved the conservation equations for the average jet flow properties. He has reported good agreement with the experimental results.

Hoch and Jiji (1981b) experimentally and theoretically studied the flow field of turbulent offset jets in an external moving stream for Reynolds number of 16000. They considered offset ratios up to 8.7 and non-dimensional freestream velocity $U_\infty < 0.3$. The theoretical study was based on the integral formulation of the basic conservation laws. They took into account both variation in pressure and radius of curvature in the recirculation region. They observed a good agreement between experimental and theoretical results. The same authors (Hoch and Jiji (1981a)) later on studied the heat transfer characteristics of turbulent offset jets for the same geometry. They provided the experimental and analytical solutions for decay of the local maximum axial temperature. They concluded that the freestream velocity had very small effects on decay of the local maximum axial temperature for the range of offset ratios and freestream velocities considered. Dakos et al. (1984) experimentally studied the flow, turbulence and heat transfer characteristics of plane and curved wall jets in an external stream for Reynolds number of 30000. The flow and wall heat-transfer measurements were done using stagnation tube and heat-flux meters (9.4 mm diameter) of the Schmidt-Boelter multiple-thermocouple type, respectively. The measurement of mean temperature was done with a $12\,\mu$m diameter chromel-alumel thermocouple sensor welded to a standard miniature hot-wire probe. The turbulence measurements were made using $5\,\mu$m diameter hot wires. They observed that flow was not self-preserving in an external stream. The position of zero-shear-stress was shifted to a point closer to the wall due to the effect of curvature. The turbulence intensity and stress in the curved outer layer were increased due to the extra strain resulting from the curvature of the wall.

1.3.3 Literature related to buoyancy-opposed jet flow

The first detailed experimental study of buoyancy-opposed jet flow is reported by Goldman and Jaluria (1986). They have carried out an experimental study of a two-dimensional buoyancy-opposed wall jet discharged adjacent to a vertical surface and buoyancy-opposed free jet to determine basic flow and thermal characteristics of such a flow. They have utilized hot-wire anemometry and thermocouples for measurements of mean velocity and temperature, respectively. They have performed flow visualization using smoke prior to the experimental study for investigation of basic nature of the flow. They have reported that the depth of penetration is mostly dependent on Richardson number (Gr/Re^2) and reduces with increase in the Richardson number. They have also found that the experimental mass flow rate increases with increase in Richardson number due to a stronger reverse flow. Kapoor and Jaluria (1989) further investigated the work done by Goldman and Jaluria (1986) to obtain the heat transfer characteristics. They have experimentally studied the heat transfer characteristics of a two-dimensional negatively buoyant wall jet flow over an adiabatic and an isothermal vertical surfaces. They have obtained the Nusselt number variation and the heat transfer to the vertical surfaces. They have reported that the rate of heat transfer and the depth of penetration both decrease with increase in Richardson number or mixed convection parameter (Gr/Re^2).

He et al. (2002) have experimentally studied the flow and thermal characteristics of a negatively buoyant wall jet which is produced by injecting hot water down one wall of a vertical passage of rectangular cross-section into a counter-stream of cold water. The measurements of local mean velocity and temperature have been carried out using the LDA and thermocouples, respectively. They have carried out the investigation for mixed convection parameter (Gr/Re^2) in the range of $0.0 - 0.052$. They have observed that the depth of penetration and lateral spread of jet reduce with increase in the Richardson number. They have also reported a concentrated mixing layer at the interface of two stream in case of stronger buoyancy influence. Craft et al. (2004) have numerically investigated the performance of different turbulence models for buoyancy-opposed wall jet flow similar to that studied by He et al. (2002). They have applied the low-Reynolds number model of Launder and Sharma (1974), the high-Reynolds number $k - \epsilon$ model and the two second-moment closures (Gibson and Launder (1978), Craft and Launder (2001)) with standard wall function and analytical wall function (AWF) (Craft et al. (2002)). They have presented vertical velocity contours, vector plots and temperature contours for isothermal flow and one of the buoyant test cases of He et al. (2002). They have reported that second-moment closure with analytical wall functions leads to a good agreement with the available results. They have also mentioned that numerical problem have presented them to obtain converged results for one of the two buoyant test case with counter-flow to jet velocity ratio 0.077 using second-moment closures.

The experimental configuration studied by He et al. (2002) is computationally investigated by Addad et al. (2004) using large eddy simulation. To quote Addad et al. (2004), “Based on the experiment of He et al. (2002) [Int. J. Heat Fluid Flow 23 (2002) 487], this flow was suggested as an “application challenge” by the power generation industrial sector to the Qnet-CFD EU network.” Addad et al. (2004) argue that numerical predictions vary significantly with the types of RANS models used. They claim that the most advanced models used by Craft et al. (2004) could yield a reasonable agreement with the experimental data of He et al (2002). However, Addad et al. (2004) have failed to mention that the real challenge is to simulate the effect of buoyancy, and Craft et al. (2004) have actually presented the validation of isothermal case and not the buoyancy-opposed thermal problem. As mentioned, Addad et al. (2004) have attempted with a hope to conform the experimental data. They have used an LES (large eddy simulation) with half-a-million nodes due to the limitation of resources. They have provided results for non-buoyant and buoyant cases. They have considered the Reynolds number 4000 and compared their computational results with He et al. (2002). However, the experimental data utilized for velocity comparisons are absent in He et al. (2002). The manuscript does not mention the Richardson number for which the computations have been carried out. So their results cannot be taken as a good comparison of He et al.'s (2002) results. They have applied two codes (Star-CD and *Code_Saturne*) to study isothermal and buoyant test cases, and have reported that both codes return satisfactory results for isothermal flow and moderately buoyant test cases.

In the companion paper, Craft et al. (2004) have classified the negatively buoyant turbulent wall jet to be a **more complex flow** than other relatively numerically amenable flows like buoyancy-modified up-and down-flow through pipes and annuli. They argue that the collision of a heated downward wall jet flow with the low-velocity upward moving cold uniform stream results in a stagnation point of the wall jet and turning of the jet upwards; this further leads to buoyant as well as dynamic influence on the stagnation point position. They further argue that a numerical prediction of the flow would require a Reynolds stress transport model (RSM) model rather than an isotropic eddy viscosity turbulence model. Craft et al. (2004) have used the new analytical wall function developed by Craft et al. (2002). Craft et al. (2004) have shown (in Fig. 4(a) of their paper) the comparison of their numerical results (using two-component-limit (TCL) model with AWF, and standard $k - \epsilon$ model with AWF) with the experimental results of He et al. (2002) and LES results of Addad et al. (2004) at downstream location 0.4 m. However, the results shown in Fig. 4(b) of Craft et al. (2004) correspond to a downstream location of 0.6 m; in the archival literature of He et al. (2002), they have provided the results up to 0.5 m. Also to be noted that, Craft et al. (2004) have carried out computations for $Re = 4000$ whereas He et al. (2002) have carried out experiments for $Re = 4754$. He et al. (2002) have carried out their experiments where ratio of counter-flow to jet velocity ratio was maintained very close to 0.077 for all the cases. Craft et al. (2004) have shown some general results for buoyant cases for counter-flow to jet velocity ratio 0.077 and 0.15. However, any deterministic comparison with He et al. (2002) for the buoyant cases is missing.

1.3.4 Literature related to turbulent natural convection flow in a cavity

The buoyancy driven flow has been the subject of extensive research in last 20 years due to its importance in many practical situations. Many researchers have investigated this problem experimentally and numerically. The experimental work of Cheesewright et al. (1986) and King (1989) at Rayleigh number (Ra) of 4.56×10^{10}, Betts and Bokhari (2000) at Rayleigh number of 1.43×10^{6} and Tian and Karayiannis (2000) at Rayleigh number of 1.58×10^{9} (square cavity) are commonly used for comparison purpose. In 1992, Eurotherm workshop was organized with the aim to provide both experimental and computational reference results for turbulent natural convection flow in a square enclosure (Henkes and Hoogendoorn (1995)). The benchmark test problem considered the buoyant flow of air in a differentially heated enclosure (aspect ratio 1) at a Rayleigh number of 5×10^{10}. The numerical reference solutions have been provided based on the computational results of 10 groups. The numerical reference results for aspect ratio 1 ($Ra = 5 \times 10^{10}$) are then compared with the experimental results provided by Cheesewright et al. (1986) for aspect ratio 5 ($Ra = 5 \times 10^{10}$). The experimental results of Cheesewright et al. (1986) for aspect ratio 5 have been chosen for comparison due to non-availability of experimental results for aspect ratio 1 and Rayleigh number 5×10^{10}. Henkes and Hoogendoorn (1995) mentioned that differences in the results for aspect ratio 1 and 5 are very small if the results are scaled with the cavity height. They further mentioned that this agreement is valid as long as vertical boundary layers are sufficiently segregated from each other (not too large aspect ratio and not too large Rayleigh number) so that the core of enclosure is stratified with horizontal isotherms. In order to avoid the relaminarization problem (laminar solution) reported with some low-Reynolds number $k - \epsilon$ models (Henkes et al. (1991), Heish and Lien (2004)), standard $k - \epsilon$ model was made mandatory for all groups. The standard $k - \epsilon$ model was not suffering from relaminarization problem however transition location was found sensitive to grid density. Henkes and Hoogendoorn (1995) suggested that the sensitivity of transition location with grid numbers can be avoided by artificially triggering the transition point with prescribed amount of turbulent kinetic energy.

The study of buoyancy driven flow in a square enclosure has been carried out numerically by Markatos and Pericleous (1984) for Rayleigh number in the range $10^3 - 10^{16}$. The flow is assumed to be laminar in the range $Ra \leq 10^6$ and turbulent in the range $Ra > 10^6$. The laminar solutions are validated with the work of Vahl Davis (1983). Two equations $k - \epsilon$ model with wall functions are used to solve turbulent transport equations. The standard gradient diffusion hypothesis (SGDH) is used for modeling of buoyant production term. They have provided Nusselt number correlations for both laminar and turbulent flows. Davidson (1990) has carried out the numerical simulation of turbulent natural convection in a rectangular cavity of aspect ratio 5 at Rayleigh number of 4×10^{10}. The author has applied two low-Reynolds number $k - \epsilon$ models. The low-Reynolds number $k - \epsilon$ model proposed by Lam and Bremhorst (1981) and the modified $k - \epsilon$ model proposed by the author (Davidson (1990)) have been applied. Both the low-Reynolds number $k - \epsilon$ models show good agreement with the experimental data. However, low-Reynolds number $k - \epsilon$ model proposed by the author predicts slightly better the location of transition. Henkes et al. (1991) have carried out the computational study of turbulent natural convection in a square cavity for air and water. The calculations have been done for Rayleigh number up to 10^{14} for air and 10^{15} for water. Three different turbulence models have been applied that include the standard $k-\epsilon$ model (in conjunction with wall function), low-Reynolds number $k-\epsilon$ models proposed by Chien (1982), and Jones and Launder (1972). They have reported that the overall heat transfer obtained from the standard $k-\epsilon$ model shows a too high prediction. On the other hand, predictions obtained from low-Reynolds number $k-\epsilon$ models are fairly close to the experimental results. They have also reported that Jones and Launder (1972) model returned laminar solution for $Ra \sim 10^{11}$ for air and $Ra \sim 10^{13}$ for water.

Davidson (1993) has numerically investigated the turbulent natural convection in a square cavity at the Rayleigh number of 5×10^{10}. He has used the standard $k-\epsilon$ model with wall functions (Rodi (1980)), the low-Reynolds number $k-\epsilon$ models of Jones and Launder (1972), the model due to Davidson (1990), and the two-layer model (Chen and Patel (1987)). He has obtained laminar solution with both low-Reynolds number $k-\epsilon$ models for both the cases i.e. when the buoyancy generation term in the turbulent transport equations is modeled with either standard gradient diffusion hypothesis (SGDH) or with generalized gradient diffusion hypothesis (GGDH) of Daly and Harlow (1970). They have applied two differencing schemes (hybrid central/upwind and QUICK schemes) that predict the similar results. Tieszen et al. (1998) have numerically investigated the tall cavity of aspect ratio 5 at Rayleigh number of 5×10^{10}. They have applied $v^2 - f$ model of Durbin (1995). The relaminarization problem has been avoided by modeling the buoyancy production term with GGDH. They have mentioned that the velocity profile in a natural convection boundary layer has some similarity with that of a wall jet flow. However, the buoyant production exceeds the viscous production $u^{'}v^{'}$ in a region between the wall and the velocity maximum. The absolute value of $u^{'}v^{'}$ in case of wall jet is nearly zero; therefore $u^{'}v^{'}$ is not correlated well with the mean velocity gradient $(\partial u/\partial y)$. Due to this reason, SGDH model breaks down in this region. However, they have mentioned that the same SGDH model is still good for modeling of turbulent heat flux in the energy equation. The inclusion of GGDH significantly improves the heat transfer characteristics and transition location with respect to the experimental results.

Peng and Davidson (1999) have numerically studied the buoyancy driven natural convection flow in a rectangular cavity at the Rayleigh number of 5×10^{10} and a mixed convection flow in a square enclosure. They have investigated the performance of low-Reynolds number $k-\omega$ model proposed by Peng et al. (1997) in detail for the case of buoyancy driven flow. They have observed that when low-Reynolds number $k-\epsilon$ model is applied to solve the buoyancy driven flow at moderate Rayleigh number ($Ra \sim 10^{10} - 10^{12}$), the model is not able to return the grid independent solution due to transition regime along the vertical walls. They have found that buoyancy source term in the turbulent kinetic energy equation exhibits strong grid dependency as it is modeled with SGDH. They have finally suggested a damping function to be multiplied with buoyancy source term to avoid grid dependency problem. Hsieh and Lien (2004) have numerically investigated the tall cavity of Betts and Bokhari (2000) at Rayleigh number of 1.43×10^6 and the square cavity of Tian and Karayiannis (2000) at Rayleigh number of 1.58×10^9. They have used variant of Lien and Leschzinar's (LL) model (Lien and Leschzinar (1999)) and two-layer approach. They have used steady RANS for tall cavity as the turbulence level in the core region is sufficiently high (v'/U_0). They have reported that the predictions obtained from LL model for the case of tall cavity show good agreement with the experimental results. However, for the case of square cavity they encountered relaminarization problem with LL model in conjunction with experimental temperature profile prescribed on the horizontal walls. To overcome this problem, they have applied two-layer model and the non-linear relation of Speziale (1987) and obtained improved predictions of mean flow and turbulence fields. However, the averaged Nusselt number is underpredicted as compared to the experimental results. Choi et al. (2004) have numerically investigated the natural convection in a rectangular cavity with $v^2 - f$ model (Durbin (1995)), modified $v^2 - f$ model (Lien and Kalitzin (2001)) and two-layer model. The authors have proposed a remedy for the numerical stiffness problem encountered with original $v^2 - f$ model. They have reported that the predictions obtained from the modified $v^2 - f$ model show good agreement with the experimental observations; however the accuracy of solution is little less than original $v^2 - f$ model.

1.4 Objectives of the present study

The low-Reynolds number turbulence models have been studied for wall jet and impingement jet flows, but the capability of low-Reynolds number turbulence models to predict complex flow field of an offset jet has not been explored. Also, very few computational works related to the heat transfer study of turbulent offset and wall jet flows have been reported in the literature. The offset jet flow is a more severe test case for validation of turbulence models as compared to the wall jet flow due to the presence of recirculation, impingement and wall jet regions. In addition, the understanding of complex flow phenomenon is also important from the fundamental fluid mechanics point of view. The problems undertaken in the present thesis are wall-bounded type in nature. The potential of low-Reynolds number models have been tested for wall-bounded forced convection, natural convection and mixed convection type of flows. The objectives of the present study are summarized below:

- To develop an in-house computer code to solve wall-bounded turbulent flows and then validation
- A comparative study of flow characteristics of wall-bounded jets using different turbulence models
- A detailed computational study of heat transfer characteristics of wall-bounded jets using different turbulence models
- Study of effects of freestream motion on flow and heat transfer characteristics of turbulent offset jet
- Investigation on the relative performance of various low-Reynolds number turbulence models for buoyancy-driven flow in a tall cavity
- Numerical investigation of a buoyancy-opposed wall jet flow