Chapter 1

Introduction

The motion of a Newtonian viscous fluid is governed by the Navier-Stokes equations. The study of unsteady viscous flow and heat transfer problems serve as theoretical models of some complicated flow problems and have several technological applications. Fluid flow and heat transfer between two parallel rotating disks are of academic and practical interest for its wide applications in rotating machineries e.g. rotating-disk contactors, gas turbines and and compressor disks and magnetic disk storage system. Heat transfer effects in lubrication have long been recognized as important in determining load bearing performance characteristics. The response of the boundary layer due to time dependent variation of the free stream has many practical applications. Specifically, it is of interest in missile aerodynamics, turbomechanics, unsteady nozzle flow etc.

The rotating flow within a cylindrical enclosure has been the subject of numerous numerical studies motivated by interest in the secondary flows generated and their consequence on various applications such as turbomachinery, chemical reactor and magnetic disk storage industries. From the literature we find that in a cylindrical enclosure with one or both endwalls rotating and a fixed sidewall, vortex breakdown phenomena occur for certain combination of aspect ratio and Reynolds number. The term vortex breakdown is associated with an abrupt change in the character of a vortex core at some axial station. Vortex breakdown in swirling flows has been the subject of much attention since it was first recognized in the tip vortices of delta winged aircraft. Vortex breakdown is an important fluid flow phenomenon because its occurrence generally limits the lift generated by low aspect ratio wings at high angle of attack. Although the major impetus for investigating vortex breakdown has been delta wing applications, vortex breakdown occurs in many other flows. For example, vortex breakdown is used favourably to improve mixing combustion using a technique known as swirl-induced combustion.

1.1 Navier-Stokes equations

The Navier-Stokes equations of motion for compressible fluid without any body force in vector notations are as follows

$$\frac{D\vec{q}}{Dt} = -\nabla \int \frac{dp}{\rho} + \frac{4}{3}\nu\nabla(\nabla.\vec{q}) - \nu\nabla\times(\nabla\times\vec{q}) \qquad (1.1)$$

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho\vec{q}) = 0$$

where $\nabla\times(\nabla\times\vec{q}) = \nabla(\nabla.\vec{q}) - \nabla^2\vec{q}$

For incompressible fluid ρ is constant and $div\vec{q} = 0$ i.e. $\nabla.\vec{q} = 0$. So the governing equations for an incompressible viscous fluid are

$$\frac{D\vec{q}}{Dt} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\vec{q} \qquad (1.2)$$

$$\nabla\vec{q} = 0 \qquad (1.3)$$

In the above equations $\vec{q}, p, \rho, \nu$ are the vector velocity field, the pressure, the density and the kinematic viscosity respectively. The total derivative notation is used above where

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + \vec{q}.\nabla$$

Thus, equation(1.2) is a vector equation expressing the change in momentum of a fluid element due to pressure and viscous force. Equation (1.3) is the continuity equation and express the conservation of mass for an incompressible fluid element. Using the vector identity

$$\vec{q}.\nabla\vec{q} = \frac{1}{2}\nabla q^2 - \vec{q}\times(\nabla\times\vec{q})$$

where $q^2 = \vec{q}.\vec{q}$, equation (1.2) may be also written as

$$\nabla P' = \vec{q} \times \vec{\omega} + \nu \nabla^2 \vec{q} - \frac{\partial \vec{q}}{\partial t}$$

Here the total pressure or head is defined as $P' = p/\rho + \frac{1}{2}q^2$ and the vorticity $\vec{\omega}$ is the curl of the velocity i.e. $\vec{\omega} = \nabla \times \vec{q}$

The solutions of the above equations become fully determined physically when the boundary and initial conditions are specified. In the case of viscous fluids the condition of no slip on solid boundaries must be satisfied i.e. on a wall both the normal and tangential components of the velocity must vanish.

1.2 Axi-symmetric Navier-Stokes equations

Although vortical flows are truely three-dimensional phenomena, many theoretical and numerical studies of these flows assume axi-symmetry in order to facilitate both analysis and computation. Under the axi-symmetric assumption, all azimuthal variations are zero, $\frac{\partial}{\partial \theta} = 0$. Written out by the components, the momentum equation (1.2) simplifies to

$$\frac{Du}{Dt} - \frac{v^2}{r} = -\frac{1}{\rho}\frac{\partial p}{\partial r} + \nu \left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u}{\partial r}\right) + \frac{\partial^2 u}{\partial z^2} - \frac{u}{r^2}\right] \tag{1.4}$$

$$\frac{Dv}{Dt} + \frac{uv}{r} = \nu \left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v}{\partial r}\right) + \frac{\partial^2 v}{\partial z^2} - \frac{v}{r^2}\right] \tag{1.5}$$

$$\frac{Dw}{Dt} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + \nu \left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial w}{\partial r}\right) + \frac{\partial^2 w}{\partial z^2}\right] \tag{1.6}$$

The total derivative, with the assumption of axi-symmetry, takes the form

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + u\frac{\partial}{\partial r} + w\frac{\partial}{\partial z} \tag{1.7}$$

The conservation of mass, as expressed by equation (1.3), simplifies to

$$\frac{1}{r}\frac{\partial (ru)}{\partial r} + \frac{\partial w}{\partial z} = 0 \tag{1.8}$$

The stream function can be defined by

$$u = \frac{1}{r}\frac{\partial \psi}{\partial z}, \quad w = -\frac{1}{r}\frac{\partial \psi}{\partial r}, \tag{1.9}$$

With the introduction of the stream function as given in equation (1.9), the conservation of mass is automatically satisfied.

The azimuthal component of vorticity vector is given by

$$\zeta = \frac{\partial u}{\partial z} - \frac{\partial w}{\partial r} \tag{1.10}$$

It may be noted that by substituting equation (1.9) in equation (1.10), a Poisson equation relating the stream function and azimuthal vorticity is found

$$\zeta = \frac{1}{r}\frac{\partial^2 \psi}{\partial z^2} + \frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial \psi}{\partial r}\right) \tag{1.11}$$

The azimuthal vorticity transport equation may be derived by taking $\frac{\partial}{\partial z}$ of equation (1.4) - $\frac{\partial}{\partial r}$ of equation (1.6). This gives

$$\frac{D\zeta}{Dt} = \frac{\partial}{\partial z}\left(\frac{v^2}{r}\right) + \frac{u\zeta}{r} + \nu\left[\frac{\partial}{\partial r}\left\{\frac{1}{r}\frac{\partial}{\partial r}(r\zeta)\right\} + \frac{\partial^2 \zeta}{\partial z^2}\right] \tag{1.12}$$

Finally, the circulation may be used to replace the azimuthal velocity. Specifically, the circulation is defined as

$$\Gamma = vr$$

the azimuthal momentum equation becomes

$$\frac{D\Gamma}{Dt} = \nu\left[r\frac{\partial}{\partial r}\left\{\frac{1}{r}\frac{\partial \Gamma}{\partial r}\right\} + \frac{\partial^2 \Gamma}{\partial z^2}\right] \tag{1.13}$$

and the azimuthal vorticity transport equation is

$$\frac{D\zeta}{Dt} = \frac{1}{r^3}\frac{\partial(\Gamma^2)}{\partial z} + \frac{u\zeta}{r} + \nu\left[\frac{\partial}{\partial r}\left\{\frac{1}{r}\frac{\partial}{\partial r}(r\zeta)\right\} + \frac{\partial^2 \zeta}{\partial z^2}\right] \tag{1.14}$$

1.3 Boundary layer theory

Suppose we have laminar flow of a fluid at high Reynolds number over a smooth solid boundary. If the boundary is at rest, then the fluid in contact with it will also be at rest. As we move outwards along the normal, the fluid will gradually increase until the free stream velocity is attained. Although this is approached asymptotically, the greatest rate of change of velocity occurs within a thin layer of fluid in contact with the boundary. This thin layer is called the boundary layer. It is a region in which the viscous forces and the inertial forces are of comparable magnitude. Outside this layer the viscous forces are very much smaller than the inertial forces. It is not possible to locate the exact points where the free stream velocity is attained. However it may be possible to define it as the thickness where the velocity is having only 1 % error with the velocity of the free stream.

The two dimensional boundary layer equations as derived by Prandtl are given by

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\frac{\partial^2 u}{\partial y^2}$$

$$\frac{\partial p}{\partial y} = 0$$

$$\frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} = 0$$

The pressure increase across the boundary layer is very small. Thus the pressure in a direction normal to the boundary layer is practically constant, it may be assumed equal to that at the outer edge of the boundary layer where its value is determined by the frictionless flow. So for the outer flow,

$$\frac{\partial U}{\partial t} + U\frac{\partial U}{\partial x} = -\frac{1}{\rho}\frac{\partial p}{\partial x}$$

where U is the free stream velocity.

Since the definition of the boundary layer thickness is somewhat arbitrary, the more physically meaningful thickness are

a) Displacement thickness (δ_1) :

The defect in the volume rate of flow caused by the action of friction is given by

$$\int_0^{\infty} (U - u)\, dy$$

If we denote this defect in the volume rate of flow by $U\delta_1$, then δ_1 is known as the displacement thickness. Thus

$$U\delta_1 = \int_0^\infty (U - u)\, dy$$

b) Momentum thickness (δ_2) :

The loss of momentum in the boundary layer is given by

$$\int_0^\infty \rho u U\, dy - \int_0^\infty \rho u u\, dy$$

or

$$\rho \int_0^\infty u\,(U - u)\, dy$$

If $\rho U^2 \delta_2$ denotes this loss of momentum, then δ_2 is known as the momentum thickness. Thus

$$\rho U^2 \delta_2 = \rho \int_0^\infty u\,(U - u)\, dy$$

$$\delta_2 = \int_0^\infty \frac{u}{U}\left(1 - \frac{u}{U}\right) dy$$

c) Skin friction :

The shearing stress on the plane boundary is given by

$$\tau_w = \mu \left(\frac{\partial u}{\partial y}\right)_{y=0}$$

the dimensionless shearing stress, is known as local skin friction co-efficient.

In some cases the thickness is considered to increase in the downstream direction and at a point after that the flow in the boundary layer becomes reversed. This causes the decelerated fluid particles to move outwards which means that the boundary layer is separated from the wall. This phenomenon is called the boundary layer separation and the point at which the boundary layer separates is known as the point of separation. The phenomenon of boundary layer separation is always associated with the formation of vortices. It occurs primarily past blunt bodies such as spheres or circular and elliptic cylinders.

The phenomenon of boundary layer separation is very much connected with the pressure distribution in the boundary layer. Since the pressure over the width of the boundary layer has the same magnitude as outside the boundary layer at any cross-section, this being one of the assumptions of the boundary layer theory, this pressure gradient is operative right through the boundary layer. Now the fluid motion in the boundary layer is determined by three factors (i) it is retarded by friction at the boundary wall, (ii) it is pulled forward by the stream above it through the action of viscosity, and (iii) it is retarded by the adverse pressure gradient $\left(\frac{dp}{dx} > 0\right)$. But due to the action of the skin friction, the forward velocity of the flow is small and so its momentum and energy may be insufficient to force its way a long distance against an adverse pressure gradient. The forward flow is then brought to rest and further on a back flow in the direction of the pressure gradient sets in, which causes the boundary layer separation. Generally speaking the boundary layer equations are valid only upto the point of separation because downstream from the point of separation the boundary layer becomes so thick that the assumptions which were made in the derivation of the boundary layer equations no longer apply. The point of separation is the point where the wall shear stress vanishes.

$$\text{i.e.,} \left(\frac{du}{dy}\right)_w = 0$$

1.4 Vortex formation

Formation of vortex takes place due to the separation of boundary layer in the vicinity of a blunt body. Let us consider the motion of a fluid past a circular cylinder. The pressure increases as the fluid particles move towards the cylinder. Since they cannot penetrate through the cylinder they move forward in contact with the cylindrical surface forming streamlines. This is a region where pressure diminishes and particles get accelerated. In the region where fluid particles are in contact with the the surface of the cylinder, a very thin layer is created where the frictional forces are actually in some role. Naturally, in this region the fluid particles will have smaller velocity. As they reach the middle of the cross section, the presence of fluid particles due to some external streams retards their flow. The pressure in this region is so high that a few of these particles get retrograde motion. This leads to a circulation, namely vortex formation.

1.5 Heat transfer

The transfer of heat between a solid body and a liquid or gaseous flow is a problem whose consideration involves the science of fluid motion. On the physical motion of the fluid there is superimposed a flow of heat. In order to determine the temperature distribution it is necessary to combine the equations of motion with those of heat conduction. It is intuitively evident that the temperature distribution around a hot body in a fluid stream will often have the same character as the velocity distribution in boundary layer flow. For example, if we imagine a solid body which is placed in a fluid stream and which is heated so that its temperature is maintained above that of the surroundings then it is clear that the temperature of the stream will increase only over a thin layer in the immediate neighbourhood of the body and over a narrow wake behind it. The major part of the heat transfer from the hot body to the colder surroundings takes place in a thin layer in the neighbourhood of the body which is called the thermal boundary layer. The problems of heat transfer in fluid flow may be classified in two categories viz (i) free convection (ii) forced convection. In the free convection flows the fluid motion is essentially caused by the buoyancy forces due to the density variation of the fluid in contact with heated boundaries.

In forced convection flows due to incompressible fluid, the buoyancy forces are negligible and the heat transfer takes place due to fluid motion only. In this case, the fluid motion is assumed to be independent of temperature and the velocity field no longer depends on the temperature field. This happens at large velocities (large Reynolds numbers) and small temperature differences.

The equation of energy of the steady flow of a viscous incompressible fluid in two dimensions is

$$\rho C_p \left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} \right) = K \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) + \Phi$$

where

$$\Phi = 2\mu\left\{\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2\right\} + \mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)^2$$

this above equation together with the Navier-Stokes equations describe the temperature field in a two-dimensional viscous flow.

The heat flux at a wall $y = 0$ is given by

$$Q = -K\left(\frac{\partial T}{\partial y}\right)_{y=0}$$

where K is the thermal conductivity, T is the temperature.

The co-efficient of heat transfer is referred to be the difference between the temperature of the wall and that of the fluid. At the boundary between a solid body and a fluid the transfer of heat is due solely to conduction. For boundary layer flow there exists a remarkable relationship between heat transfer and skin friction.

1.6 Numerical method of solution

The governing Navier-Stokes equations under specified initial and boundary conditions could not be solved analytically in many cases. In the present study numerical solution has been attempted using finite difference techniques. While using finite difference technique the convective terms of the Navier-Stokes equations have to be discretized with particular care. Generally, diffusive terms do not present any difficulty. Three-point second order accurate central difference formula is widely used to represent the diffusive terms. If second order accurate central differencing technique is used in the convective terms, difficulty arises in the stability of the scheme, particularly at high Reynolds number; since the domination of the flow due to viscous terms reduces. On the other hand, introduction of the upwind finite difference approach to discretize the convective terms, permits computations of complex recirculating flows even at high Reynolds numbers.

The main disadvantage of pure upwinding is the inaccuracy arising out of the spurious numerical diffusion for flows skewed to the grid system (Patankar 1980). A series of high order and composite upwind schemes have been developed by Leonard (1979) and Zhu (1991) respectively.

Higher-order accurate discretization can also be obtained by so-called compact differencing techniques. These consider not only the variable itself as unknowns but also its derivatives as well. The compact methods cause no instability but they are much more complex to implement. A compact difference scheme which needs only three nodes to obtain fourth order accuracy has been developed by Hirsch (1975), Adam (1977) and Peyret and Taylor (1983). The composite schemes use an appropriate combination of low and high order upwind and central difference schemes based on a physical criteria of convection boundedness (Gaskell et al. 1988). Many discretization schemes to simulate Navier-Stokes equations have been reported in the literature namely, Noye (1978), Correa and Shyy (1987), Fletcher (1988), Hirsch (1990). The second order upwind scheme has received continued attention and has been investigated by Atias et al. (1977), Shyy (1985), Shyy et al. (1992). In order to eliminate the deficiencies of first order upwind finite difference schemes several modifications of the scheme have been proposed. Among them, the skewed upwind finite difference scheme (Stubley et al. 1980) and the quadratic upwind finite difference scheme, QUICK (Leonard 1979) have gained popularity.

Based on both linear analysis and model problem testing, Shyy (1985) demonstrated that the second order upwind scheme, although not free from difficulties and inaccuracies, is generally more satisfactory than several other alternatives proposed in the literature. Vanka (1987) on the other hand, found that the second order upwind scheme does not yield satisfactory performance, both in terms of numerical accuracy and computational stability. Durst and Pereira (1983) carried out comparative computation with four different finite difference formulations involving (i) upwind finite difference scheme (UDS), (ii) hybrid central/upwind finite difference scheme (CUDS), (iii) quadratic, upstream weighted finite difference scheme (QUDS) and (iv) hybrid central/skewed upwind finite difference scheme (CSUDS). All the four schemes were applied to the two dimensional steady laminar separated flow problems. Among other finite difference methods, the ADI (Alternating Direction Implicit) method by Peaceman and Rachford (1955) and Douglas (1955), Multigrid method by Brandt (1977), Jiang et al. (1990), the FFT (Fast Fourier Transform) method by Schumann and Sweet (1988) and Fractional-Step method by Kim and Moin (1985) have received much attention in recent times.

Discretization of the governing equations yields a system of algebraic equations which has to be solved by standard procedures. For the solution of such algebraic systems, iterative methods are frequently used (Varga 1962, Golub and Vanloan 1989). Most of the direct methods involve much effort and storage. It suffices here to mention that the best known amongst them are viz., the point Gauss-Seidal iteration (GS), Successive over-relaxation (SOR), line Gauss-Seidal iteration (LGS), line successive-over relaxation (LSOR).

It is well known that the incompressible continuity equation introduces difficulties into the solution procedure for Navier-Stokes equations in primitive variables, in that a main dependent variable can no longer be identified in the equation. Harlow and Welch (1965) proposed the staggered grid variable concept, MAC (Marker and Cell) scheme which proved over the years to be a robust technique to solve the coupled equations. The derivation may be achieved either by using the parent differential equations, Roache (1985) or by invoking the finite difference analogues of the three equations directly as in Patankar (1980), Peyret and Taylor (1983). Several well known computer codes were developed e.g. SIMPLE by Patankar and Spalding (1972), SIMPLER by Patankar (1980), SMAC by Amsden and Harlow (1970), SOLA by Hirt et al. (1975), PISO by Issa (1985). The projection method of Chorin (1968), the finite volume method with staggered and collocated grids by Peric et al. (1988), the method of Fractional steps by Yanenko (1971), Temam (1969) are also popular for the solution of unsteady incompressible Navier-Stokes equations.

1.7 Scope of the thesis

The thesis presents theoretical investigations on some unsteady viscous flow and heat transfer problems.

A realistic approach to study many natural flows is to take the unsteady aspect into consideration. Due to complexity in the non-linear governing equations in complex flows analytical solution is not possible. Efficient numerical namely , Crank-Nicolson implicit method, Alternating direction implicit method (see Appendix E and Appendix F) have been employed to study the complicated flow problems. Numerical stability and order of accuracy have also been studied. The computed numerical results have been compared with the available previously published numerical and experimental results for all the problems. The results are found to be in excellent agreement. All computations have been made using double precision. Since all the problems have been solved numerically, basically no assumption has been carried out. The thesis consists of eight chapters.

The problem of unsteady viscous flow and heat transfer due to impulsive motion of a cone has been studied in Chapter 2. The development of boundary layer past a semi infinite flat plate impulsively set into motion was studied by many investigators. A semi-similarity transformation has been applied to the governing equations. The resulting set of parabolic type equations describing the transition of boundary layer from the initial steady state to terminal steady state is found to change to elliptic type in the solution domain and the usual forward integration methods fail. Thus for computation we have used a second order upwind difference scheme in the time-like variable. Newton's linearization technique has been used to cope with the non-linearity. The temperature distribution and heat transfer have been obtained for the cases of constant wall temperature and constant heat flux at the wall. Numerical results show that the magnitude of dimensionless velocity and that of temperature decrease with the increase of time-like variable (ξ) for constant wall temperature case and they increase with the increase of ξ for constant heat flux case. Further with the increase of m, the Falkner Skan parameter, which depends solely on the cone half angle, the magnitude of skin friction and that of wall heat transfer increases. It has been found numerically that flow separation does not occur for the unsteady flow past a cone for $m \geq -0.275$.

Chapter 3 deals with the problem of unsteady rotating flow of a compressible fluid over a finite disk. A continuous time dependent variation of the free stream azimuthal velocity has been considered in the present study. Both accelerating and oscillating type of free stream velocity distributions have been considered. Due to reversal in radial velocity over the inner half of the disk, the boundary layer equations become a type of time dependent singular parabolic equation and its solution requires the conditions over all the boundaries. Finite-difference scheme using space-centered differences in the axial direction and upwind differences in the radial direction have been used for solving the governing equations. Crank-Nicolson implicit scheme has been used for time-wise discretization. The velocity profiles near the edge of the disk are found to be non-oscillatory but from the mid-radius and onwards radial and tangential velocities exhibit oscillation about their free stream values. Magnitude of skin friction parameter increases with time in an accelerating free stream but they exhibit oscillation in an oscillating free stream. Skin friction parameters increase with the increase in disk temperature.

The unsteady flow and heat transfer of a viscous fluid film squeezed between two parallel rotating disks have been studied in Chapter 4. The disks are allowed to rotate with different time dependent angular velocities and the upper disk is made to approach the lower one with a constant speed. The angular velocities of the disks have been taken to be either accelerating or decelerating with different values of upper and lower disk rotation ratio. A fourth order accurate compact finite difference scheme using Hermitian relations has been used for solving the governing parabolic partial differential equations. The load bearing capacity increases with the increase of the rate of squeezing but it increases considerably as the distance between the two disks decreases. At a fixed rate of squeezing and Reynolds number (based on angular velocity of the lower disk), the load decreases when the upper disk is in counter rotation than the case in which the upper disk is non-rotating. Increase in squeezing rate and angular velocity of the lower disk increases the torque exerted by the fluid on the disks. The magnitude of heat transfer rate at both the lower and the upper disks increases with the increase of time and also with the increase of the rate of squeezing. Heat transfer rate also increases with the increase of angular velocity of the lower disk.

The flow inside a cylindrical container induced by the impulsively started uniform co-rotation or counter rotation of the top-bottom endwalls with fixed sidewall has been investigated in Chapter 5. The unsteady axi-symmetric Navier-Stokes equations describing the flow have been expressed in vorticity stream function form. The ADI scheme is used to solve the azimuthal vorticity transport equation and circulation equation. For large values of Reynolds number an upwind differencing in the spatial derivatives for convective terms has been used. A fourth order accurate compact difference scheme has been used to solve the Poisson's equation for stream function. Numerical result shows that steady state motion arises after a transient period. Here the values of aspect ratio (A) are taken to be 0.5, 1 and 1.5. For the case of co-rotating endwalls at lower values of Reynolds number Re (< 1000), a small recirculation region occurs close to the point of symmetry when the aspect ratio A is less than or equal to 1. But when the aspect ratio exceeds 1, two stagnation points are found to occur one at the upper endwall and the other on the shroud. Associated with these stagnation points is the large counter rotating flow region in the upper half of the domain. For the case of counter rotating endwalls, two regions of counter rotating flows are observed with the dividing streamline lying entirely between the sidewall. When the disks rotate with equal and opposite angular velocities a shear layer develops along the midplane. When Reynolds number exceeds a critical value the stream surfaces show undulation about the axis of symmetry. A closed region of recirculating fluid appears on the axis of rotation. This phenomenon is known as bubble type vortex breakdown. No breakdown bubble is found to occur for the flow due to counter rotation of top-bottom endwalls.

Chapter 6 deals with the effect of slight co-rotation of the top-bottom endwall on vortex breakdown phenomena in a cylindrical enclosure. In the present study, the endwalls are impulsively set to rotate with uniform but different angular velocities. The unsteady Navier-Stokes equations have been solved using the same method which has been described in Chapter 5. The numerical results for fixed aspect ratio $A = 1.5$, show that beyond a critical value of the Reynolds number ($Re \geq 1600$) for a particular value of ϵ (the ratio of the angular velocities of the top and bottom endwall) waviness in the stream surfaces of the central vortex occurs and a single breakdown bubble appears on the axis of rotation. The size of the bubble increases with increase of Reynolds number. Due to small co-rotation ($\epsilon = 0.1$) of top-bottom endwall the shape of the breakdown bubble remains unchanged but it shifts its position towards the faster rotating endwall. It is found that a co-rotation of the top endwall upto a certain value of the top-bottom endwall rotation ratio ($\epsilon \leq 0.5$), decreases considerably the critical value of Reynolds number for the onset of vortex breakdown. The region