CHAPTER II
LITERATURE SURVEY AND
RESEARCH METHODOLOGY

A detailed review of literature appropriate to the research problem is carried out in this chapter. The observations and outcomes of the allied works would be helpful for understanding the involved phenomena better. Also they lay the foundation for the current research by developing the research scope and methodology.

2.1 Flow characteristics of annular and simple labyrinth seals

A sound knowledge of the leakage flow characteristics of annular and simple labyrinth seals is essential for accurately predicting the pressure drop across them.

2.1.1 Flow through annular seals

While analyzing fully developed turbulent flow through annulus, Jones and Leung [2] observed flow instabilities in the form of double helical flow pattern appearing to degenerate into swirls. This caused higher friction to happen at Reynolds numbers around 1550. For experimental investigations on flow through any non-circular geometry, sufficient pressure drop data on laminar flow through annuli should be obtained to insure the adequacy of experimental procedure. This is followed during the entire course of the current work.

Villasmil et al. [16] observed that Hirs’ bulk flow theory used to determine annular flow leakage and dynamic coefficients, relies on empirical friction factor correlations. Due to this, while leakage is well predicted, the dynamic coefficients are not. Water flow over deliberately roughened seal surfaces was simulated using commercial CFD code Fluent and found an increase of friction factor with higher clearance. This is because of the development of a high static pressure in the trailing face of each roughness cavity while the wall shear stresses on the smooth land
played a secondary role. In a certain Reynolds number range, the maximum friction factor observed on a specific roughness pattern size is independent of the actual clearance. This is referred to as the friction factor-to-clearance indifference behaviour. This phenomenon is found to be related to the roughness of cavity and its length to clearance ratio. The authors concluded that a stable mean flow recirculation zone within each cavity could lead to larger friction factors and the results are applicable to predictions for cases with no shaft rotation.

Lessen [35] performed an analytical study of flow in a plain annular seal. He modeled the axial component of flow and found that vortices often trigger transition to turbulence in the boundary layers. It is also found that the azimuthal circulation for the static cases is the same for both centered and eccentric rotors. It is further observed that the radial reaction force is inertial and induces instabilities in the flow, while the azimuthal reaction forces are purely viscous.

Yowakim and Kind [36] measured total and static pressures and used hot wire anemometry to measure the mean velocity and normal Reynolds stresses in a non-rotating concentric annulus. They studied flow with inlet swirl angles from 0° to 45° and found higher shear stresses on the outer wall than the inner wall and the existence of approximately isotropic flow.

Kind et al. [37] applied the law of the wall mixing length relation to non-rotating, swirling flow through an annulus. The expressions for the logarithmic portion of the law of the wall were derived for the axial and tangential velocity components after the appropriate shear-velocity scale for each component direction is identified. By comparison with the previous data, the derived profiles were shown to agree well near each wall. The success of the expressions validated the mixing length model used in deriving them and indicated that the velocity gradient at any height \( y^+ \) in the near wall region is governed by the wall shear stress rather than the local shear stress. In the current work, these would be taken into account while undertaking CFD analysis on the seals.
Simon and Frêne [38] presented an analysis for incompressible flow in annular pressure seals to calculate the static and dynamic characteristics of plain annular seals. To evaluate the effects of inertia forces in the film, tapered geometry and rotor misalignment were taken into account. Their model is based on the Navier Stokes equation, the continuity equation and a turbulence model. The inlet boundary conditions defined the initial swirl and the pressure drop due to the fluid acceleration.

Morrison, et al. [39] experimentally measured the velocity field and full Reynolds stress tensor in a centered plain annular seal and compared simulation results using a Laser Doppler Anemometer (LDA) system. It was found that positive radial velocities at the inlet region which were attributed to sudden swirl generation. It was observed that the development of anisotropic turbulence as the flow moved downstream.

Arghir and Frêne [40] presented a calculation of forces and moments in misaligned plain annular seals operating in centered positions and turbulent flow regimes. The method is based on the numerical integration of the perturbed averaged Navier-Stokes equations so that it had the capacity to treat recirculating flows in seal grooves. The boundary conditions were obtained from the no slip conditions and from the logarithmic law. Validation of the method for a straight seal was made using the experimental results of Kanemori et al. [8]. Force and moment for a four-grooves on stator labyrinth seal were also discussed. The effect of groove depth on the radial and azimuthal force and momentum exerted on the stator was studied. The effect of a positive radial momentum was found to increase the misalignment angle. Increasing the groove depth decreased this effect.

To better realise the physics of leakage control and rotodynamic stability of honeycomb-stator gas annular seal, Chochua et al. [41] developed a computational capability for low Mach number, compressible, turbulent flow between two straight honeycomb plates. It was found that the flow does not reach the fully developed condition until about 30-40 % of the channel length and the periodic treatment of
boundary condition is valid only after this length. Yet, the repeatability of the flow field could accommodate adoption of periodic treatment also. The pressure/form drag dominates the wall shear stress. Strong pressure gradients in the vicinity of downstream walls are balanced by enhanced shear stresses from the mixing process explaining the better leakage control of these seals compared to smooth wall seals.

2.1.2 Flow through simple labyrinth seals

Labyrinth leakage flow resistance is affected by several geometrical dimensions of the seal in a complicated manner. Some investigators had initiated studies on labyrinth seals with flow field visualisation in them. To achieve enough resolution on very small features of the flow fields, the seal geometry was scaled up to enable clearer visualisation at critical points in the seal [11].

El-Gamal et al. [17] investigated the laminar incompressible flow and the performance of different types of labyrinth seals under both stationary and rotating conditions. A three dimensional numerical model to analyse labyrinths shapes of small height to radius ratio was formulated. Shaft rotation was found to be beneficial only for an up-the-step seal and had no effect on grooved shaft and grooved casing seals and had an adverse effect on the down-the-step seal. It was also found that a small clearance at the entrance made improvement in the performance of the grooved shaft and down-the-step seals depending on the value of the height to width ratio.

Stocker [23] had utilised flow visualisation techniques for advanced seal geometries. Several stepped labyrinth seal designs were tested by using three different testing and evaluation phases. The first phase involved a larger scale 10:1, planar water tunnel with transparent plastic walls for the observation and photography of air bubbles. Promising enough seals were selected on the basis of bubble concentration density, recirculation velocity and measured leakage rate. In the later phases, visualisation at actual scale and experiments were carried out.
Brownell et al. [42] analysed leakage resistance and flow visualisation data in an integrated fashion to get an improved understanding of the mainstream path and turbulence resulting within the cavity. Rhode et al. [43] used VCR recordings to study which change in flow field accompanied a significant change in leakage resistance due to geometrical change. Both white pliolite particles and a plastic/aluminum mixture of silver glitter were tested as flow tracer particles with the silver glitter giving the best results.

Computational methods could also be employed for getting leakage resistance data and enhanced understanding of the responsible mechanisms of the seal flow fields [44]. The visualisation studies brought out the fact that within the labyrinth seal the flow proceeds like a series of adiabatic throttling processes since windage heating had been found to be low. Pre-swirl had been found to have destabilising effect on the flow and the leakage flow enhances frictional drag affecting power absorbed.

Several researchers have devised simple algebraic models for the estimation of pressure drop for incompressible flow through labyrinth seals. Some of them are outcomes of experimental data correlations while some are purely analytical expressions involving friction factors. For straight through labyrinths, Jerie [11] took into consideration the effects of discharge factor, diffusion angle of the stream, turbulent mixing and vorticity in between seal constrictions. The flow on a macroscopic scale was examined without considering compressibility.

Later, Vermes [13] determined the flow parameters both analytically and experimentally. Bilgen and Akgungor [45] had developed an analytical model assuming Couette and Poiseuille flows in the tangential and axial directions respectively. Nikitin and Ipatov [46] had employed labyrinth seals to produce less leakage than plain annular seals for a given clearance and length. The frictional losses were approximated through theoretical analysis and experimental data correlation. The leakage rate, pressure drop and friction coefficients varied as a
function of clearance, pitch, tooth height, tooth width, etc. Such studies [14,46] had mainly resulted in empirical leakage prediction techniques.

Han [47] estimated the shear stress along the free shear layer by considering the leakage flow energy consumed in driving the cavity vortex. The wall shear stresses were evaluated using Prandtl's flat plate boundary layer solution. Hass and Muller [48] showed that labyrinth seals can be effectively sealed even under severe liquid spattering conditions and gave an outline for a mathematical leakage flow model.

Konnur and Rammohan [49] observed that in rotary labyrinth seals apart from the normal function, the centripetal force also plays a role in the leakage flow. For static labyrinth seals, the effect of windage losses is zero. The pressure drop created by the seal geometry alone decides the leakage flow. The authors evaluated pressure drop at the sealing part by considering the flow as annular taking place between smooth and rough pipes. At higher values of Reynolds numbers, the effect of increasing the cavity depth is less pronounced.

2.2 Computer simulation and experiments on annular and labyrinth seals

The major advantage of applying computer simulation to flow through seals lies in the ability of CFD to accurately predict the flow through seals when analytical formulations are unavailable. Also, seal performance can be investigated before the seal is even fabricated. The performance includes the leakage, pressure drop, cavitation coefficients etc. The first step is to obtain the flow field. The simulation of different plain annular and labyrinth seals are reviewed hereunder.

2.2.1 Studies on annular seals

Morrison, et al. [50] used the Reynold's averaged Navier-Stokes equations for flow through annular seal and conducted experiments. Three pre-swirl angles of -45°, 0°, and +45° were considered. The simulation showed good agreement in the mean velocities. However, the numerically predicted Reynold’s stresses were underestimated compared to the experimental results.
Thames [51] performed computer simulation using the CFD code Fluent to measure flow fields in an annular seal and used 3D laser-Doppler anemometer system for benchmarking purposes.

Hatfield [52] presented a similar simulation of the effect of upstream swirl upon the flow field in a centered plain annular seal. The Starzone hydro-code developed by Titan Corporation is employed. The code solves the Reynolds averaged Navier-Stokes equations. A second order closure turbulence model was used including differential equations for the six Reynolds stresses and an integral turbulent length scale.

Athavale et al. [53] presented a simulation of flow through a whirling plain annular seal geometry and compared the results with experimental data. The simulation is done using the 3D CFD code Sciseal. A standard $k-\epsilon$ model with wall functions is used to treat the turbulence. Experimentally measured values of the flow parameters were used to specify the seal inlet and exit boundary conditions. The computed flow in terms of the velocity and pressure is compared with the experimental measurements inside the seal and revealed good results for the axial velocity, but only fair results for the radial and tangential velocity and the pressure on stator.

### 2.2.2 Studies on simple labyrinth seals

Stoff [54] studied the incompressible flow in a labyrinth seal. The flow was computed using a $k-\epsilon$ turbulence model with a pressure-velocity computer code solution of the Navier-Stokes equation in order to explain the leakage phenomena due to the mean pressure gradient. The main stream in the circumferential direction was found inducing a secondary mean flow vortex pattern inside the labyrinth cavities on the surface of the shaft. The finite difference solution was compared with measurements obtained by a back-scattering laser-Doppler anemometer. The simulation produced satisfactory results for the problem of leakage in the labyrinth.
seal. The influence of turbulent fluctuations on the mean momentum transport was sufficiently well represented by the energy dissipation equation of the turbulence model.

Wittig et al. [55] observed dependence of flow coefficients and friction factor on the scale effects arising out of dynamic similarity. Conditions in the present work are set to alleviate similar problems.

Morrison and Daesung [56] experimentally investigated a stepped labyrinth seal to determine the effects of pressure ratio, number of teeth, shaft speed and tooth/step location upon the leakage of water through the seal. The dependence of the flow coefficient upon the number of throttlings and pressure ratio are similar to those for straight through labyrinth seals. Axial pressure distribution measurements show that when the teeth are centered on the step, the pressure drop from cavity to cavity is almost uniform. This aspect along with decreased kinetic energy carry over was speculated to be the cause for the enhanced performance of this type of seal. The flow coefficient decreased as the number of throttlings increased, making the latter as the most important factor in designing a labyrinth seal since it has the most significant effect.

The leakage resistance mechanism is basically related to the intense turbulence generation and dissipation near the cavity inlet and the turbulent momentum exchange near the various recirculation zones. In order to show the effects of the various design dimensions, Demko et al. [57] dealt the dimensionless inlet to outlet bulk pressure drop for each cavity through a minor loss coefficient which is commonly used to represent mechanical energy losses in valves, elbows, etc. for flow through pipes.

Ha and Childs [58] designed a flat plate tester to measure static pressure drops, flow rates and temperatures for flow passing through two closely spaced honeycomb surfaces. They found that the multiple small cells of honeycombs created multiple vortices thereby reducing leakage in a flat plate tester.
Rhode and Sobolik [59] employed the QUICK convective differencing scheme to eliminate false diffusion in the prediction of subsonic compressible flow in a rotodynamic labyrinth seal. This scheme served to enhance the designer's insight and facilitated the development of friction factor type flow models. This served as a motivation during the development of an analytical model in the present work.

Nordmann et al. [60] presented a model for a circumferentially grooved annular seal. The model accounted for the difference in friction factor in the circumferential and axial directions due to the grooves and used an average groove depth to account for the circumferential flow in the grooves. The simulation was within 20% of the experimental results for direct stiffness. Entry swirl of 10% to 30% of the rotational speed had to be taken into account to fit theoretical values to the best results.

Kilgore and Childs [61] obtained predictions of leakage flow and rotodynamic coefficients based on the above grooved seal model [60] for six circumferentially grooved liquid seals. Comparing them with measured values showed that friction factors in grooved seals with orbiting rotors showed a poor correlation. The friction factors of the model increased with increasing rotor speed, while the measured friction factors have mixed speed dependency at lower Reynolds numbers and a definite decrease with increasing rotor speed at higher Taylor numbers. The prediction of rotodynamic coefficients was reasonable, under-predicting the equivalent linear stiffness coefficient by 40% and over-predicting the equivalent linear damping coefficient by 10%. Their present study was oriented against directly correlating rotary seal results.

Rhode et al. [62] developed a TEACH type finite difference computer code to calculate the pressure drop for incompressible leakage flow through simple and stepped labyrinth seals, without requiring the estimation of the often uncertain kinetic energy carry over. Their method alleviated the convergence difficulty which arises
upon implementation of the QUICK scheme.

Zimmermann and Wolff [63] worked out a correlation method based on the knowledge from flow visualisation tests and numerical flow field calculations. Accordingly, by dealing the first cavity separately a good agreement with test results was achieved.

Wittig et al. [64] used a numerical procedure based on the time averaged Navier-Stokes equations with the turbulence characteristics described by a $k-\varepsilon$ model for the computation of the two dimensional flow fields. They compared their computational results to pressure distributions measured along a planer two-dimensional stepped labyrinth seal. Both straight through and stepped labyrinth seal designs were studied and measurements were made with air as the working medium. The mass flow rate through the seal was determined using an orifice meter. Pressure measurements were made along the length of the seal. The calculated velocity field for the stepped seals predicted the presence of large vortices within each labyrinth chamber. Comparisons of the measured and predicted discharge coefficients showed good agreement. The pressure drop through stepped seals exhibited a reduced dependence on the clearance as compared to straight through plain annular seals.

Morrison et al. [65] measured the velocity field and full Reynolds stress tensor in a centered labyrinth seal using water as the working fluid. Swirl vanes were placed upstream of the seal to produce positive, negative and no pre-swirl. The seal was a seven-cavity tooth on rotor labyrinth seal manufactured of acrylic, had a polished smooth surface and was coated with an anti-reflecting material. There were eight straight teeth with a thickness of 1.524 mm and a height of 3.048 mm. The tooth pitch was 4.572 mm. The flow inside the seal was measured using a 3D laser-Doppler anemometer system. The axial velocity and radial velocities were minimally affected by pre-swirl. The tangential velocity, both in the clearance region and the seal
cavities on the rotor were greatly altered by the pre-swirl. By applying negative pre-swirl, the tangential velocity was suppressed, even in the seventh cavity. The turbulence levels decreased as the pre-swirl varied from negative to positive.

Rhode and Nail [66] developed a computation for cavity-by-cavity flow development in generic labyrinth seals. They improved a developed version of a swirl-flow finite difference computer code and employed it in predicting the compressible flow of air through generic labyrinth seals. The results for straight through seal with teeth on stator and teeth on rotor were given. Teeth on rotor seals give greater leakage resistance than the equivalent teeth on stator design. Also, they exhibit a greater swirl velocity development.

Rhode and Hibbs [67] presented a three-dimensional computation of rotodynamic force distributions in a labyrinth seal with air as the working fluid. They applied a coordinate-transformation to the Reynolds time averaged Navier-Stokes equations. The SIMPLER algorithm with QUICK differencing and a high Reynolds number $k-\varepsilon$ turbulence model were used to compute the complex turbulent flow field. The swirl development was slightly higher for thicker teeth and the second cavity yielded an increase in turbulence energy and length scales over the first cavity.

Mukhopadhyah et al. [68] predicted viscous flows in complex geometries including labyrinths by dividing the flow domain into non-orthogonal control volumes, isoparametrically mapping on to standard rectangular cells. They accomplished the evaluation of pressure distribution through an iterative correction of pressure and velocity.

Rhode and Guidry [69] carried out a parametric study of geometrical effects on labyrinth seal leakage resistance using a Navier-Stokes finite difference code and a new term effective cavity friction factor. They found that it was the sharp streamline curvature at high velocity slightly upstream of the vena contracta which provides the primary intense turbulent friction.
Marquette et al. [70] developed an extended three-control volume theory for circumferentially grooved liquid seals. They entertained the transfer of momentum terms and considered diverging flow in the flow-through section within a seal groove. The modifications in the seal parameters induced a correct evolution of the rotodynamic coefficients. Direct and cross coupled stiffness coefficients were slightly under-predicted while the leakage flow prediction was good.

Prasad et al. [71] carried out computational and experimental investigations on static straight-through labyrinth seals over a wide range of pressure ratios and for clearance values of 0.2 mm, 0.36 mm and 0.6 mm. The Fluent simulation results agreed with the experimental data within 8.6 %. The behaviours of streamline and Mach number contours had been presented. It was found that turbulence increases energy lost in the seal and decreases leakage.

Asok et al. [72] developed a FEA based computer program for the solution of flow field in lid driven cavities having different aspect ratios. The behaviour of the vortices developed inside the cavities brought out by the centre line distribution of velocities favored square geometry in the aspect of incurrence of higher pressure drop. In fact, the first circular-grooved labyrinth seal investigated in the present work has square cavity. Flow reversal near the bottom of the cavities was noticed at higher values of Reynolds numbers.

### 2.2.3 Studies on advanced labyrinth seals

Towards creating increased flow resistance, the advanced labyrinth seals usually have their profiles complicated in several ways [73]. During the development of their TEACH computer program, Gosman and Pun [74] had illustrated the important role of vortices in dissipating flow energy in recirculating flows. The advanced seals make all out efforts in strengthening their zones of recirculation. Meyer and Lowrie [75] ran leakage tests over a wide range of pressure ratios and clearances for several advanced types of seals having slant cavities and honeycomb surfaces. Their results showed good improvement over simple labyrinth profiles, but
it was cautioned that discharge coefficient of a seal soared high when a second seal was added downstream. This aspect prevented the present study from going into consideration of different labyrinths kept in series.

Stocker [76] had undertaken studies to develop advanced labyrinth seal designs, reporting minimal effect on leakage due to rotation speeds up to 235 m/s. The seals were incorporated with geometric features capable of substantially increasing the seal cavity turbulence. Implementation of honeycomb lands reduced leakage by 24% at 0.51 mm clearance, but it was found to increase leakage as clearances decreased to 0.13 mm. While the effects of some geometric parameters can be analytically calculated, the interactions of the numerous geometric parameters are complex and do not lend themselves to analytical methods. Water tunnel flow visualisation was used for preliminary evaluation of the advanced candidate labyrinth seal designs. Taking cue from this work, the present research is started with visualisation studies and continued with extensive computer simulation of leakage flow.

Wyler [77] developed a double labyrinth seal geometry for use in aircraft fuel flow meters. It reduced leakage to 40% of an equivalent annulus while the power dissipation for both is about the same. Childs et al. [78] showed that interlocked seals leak substantially less than simple labyrinth seals and the destabilizing forces were lower. Absence of any prediction method precluded discussions between theory and experiments.

Hart and Brown [79] studied the use of swirl to control the vibration behavior of rotating machines. A nozzle pointing backwards relative to the shaft rotation was used to inject the flow in the seal with a backward swirl. It was concluded that a deliberate injection of negative swirl in an annular space has potential for reducing forced vibration near resonance. This study promoted promotion of designs having swirl effects in the current research.
For evaluating the pressure distribution of the helical-grooved labyrinth seal, Heital [80] developed a finite element based numerical calculation, however by neglecting the inertial forces. They investigated rectangular cavities and concluded that multiple start threads would augment pressure drops even higher.

2.3 Optimisation of labyrinth seals

The differing sealing efficiencies prove the existence of optimal labyrinth seal geometry. There is no general tool available for optimising labyrinth seals. CFD has been successfully applied to investigate selected labyrinth seal configurations [74]. CFD enables automated shape optimisation of labyrinth seals, like it has been frequently applied to determine optimal shaped airfoils and diffusers.

Rhode et al. [81] presented a manual approach to labyrinth seal optimisation on the basis of the ability of CFD to investigate selected labyrinth seal configuration. They systematically varied the knife pitch, fin width, step height and clearance of a stepped labyrinth seal. In all the cases, the step was positioned at the same relative axial position. CFD was used for evaluation of the seals. The determined optimal seal was finally tested experimentally leading to a distinct leakage reduction in comparison to a baseline design.

Aboulaich et al. [82] carried out optimisation studies for their single cavity straight labyrinth seal using a finite element code. They found an optimal shape for incompressible laminar flow conditions with minimal leakage.

Schramm et al. [83] presented an optimisation method to minimise the leakage through a three finned stepped labyrinth seal. They coupled a program for a parameterised, automatic grid generation with a commercial CFD flow solver and an optimisation algorithm. Since standard optimisation strategies like gradient based methods are mostly trapped to local optima, they applied simulated annealing which is a heuristic method allowing the finding of global optima of the evaluated functions. The position and height of the labyrinth step were the variable design parameters. The optimal seal geometry had one step distance to the upstream knife of 85 % of
the knife pitch. The optimal step height equalled five times the seal clearance. The complete optimisation procedure carried out on two processors took seven days to complete. The discharge coefficient of the optimal seal showed a leakage reduction of about 10%.

Artificial Neural Network (ANN) employs artificial neurons to perform complex operations using already gathered knowledge [84]. It can perform nonlinear mapping using randomly assigned weights. Ohdar and Pasha [85] gave an ANN approach for predicting powder metallurgy process parameters. Their ANN model is based on three layer neural network with back propagation learning algorithm. The training data were collected by an experimental setup. The predicted value of density coincided well with the experimental value. The ANN approach not only helped in the reduction of the experimentation required, but also reduced the problems associated with empirical models involving the evaluation of many constraints. The present study also contemplates the use of ANN to optimise the simple labyrinth seal.

Another methodology of optimisation is also studied. Based on the Taguchi quality design method and the analysis of variance, Liao et al. [86] determined several significant factors involved in the machining of wire electrical discharge machining. By means of regression analysis [87-89], mathematical models relating the machining performance and machining parameters were established. The mathematical models derived by the regression analysis are sufficiently precise to represent the real machining process. Based on the mathematical models developed, an objective function under the multi-constraint conditions was formulated. The optimisation problem was solved by the feasible direction method using a programmed software yielding the machining parameters corresponding to the shortest machining time satisfying the requirements of accuracy and surface roughness. The present work would also follow the above methodology except that a genetic algorithm [90] based software Genehunter would be employed to solve the
optimisation equation. More discussion on genetic algorithm is presented later where applicable.

2.4 Computer simulation of discharge and cavitation in valves

Industrial fluid transiting systems can experience severe low frequency vibration caused by cavitating liquid flow resulting in heavy noise also. Several means of cavitation predictions are available to design systems against such risks [91]. Hassis [92] conducted experimental studies on the effects of cavitation in butterfly and monovar valves with focus on unsteady pressure fluctuations. Cavitation induces a decrease in resonance frequencies. A non-coupling between upstream and downstream of the pipe also was observed.

Vaughan et al. [93] performed experimental visualisation and simulation of liquid flow through poppet valves and found qualitative agreement between the flow patterns. They noted quantitative errors in the prediction of jet separation and reattachment and attributed the same to the limitations of the upwind difference scheme and representation of turbulence by $k$-$\varepsilon$ model. Hence, the present work would employ the RNG turbulence model.

Rahmeyer et al. [94] reduced cavitation in a control valve through the use of a valve trim. It enabled right angle turns and multiple flow paths for the liquid and replaced overall pressure drop by multiples of smaller pressure drops. Each individual pressure drop is small enough that any cavitation produced will be minimal. Each flow passage was made to continuously increase in flow area so that the velocities are continuously reducing, thus minimising the potential for damaging pressure recovery. The incipient cavitation was measured by a stethoscope. Since machining of labyrinth grooves on a conical valve would lead to the above situation, the idea of Labyrinth Conical Valve (LCV) has been mooted for the present investigations.

Merati et al. [95] used dynamic pressure transducers to find the Strouhal
frequency, LDV to measure mean velocity and turbulence magnitude, high speed video photography for flow visualisation and Fluent code to develop a two dimensional fluid dynamics model in the investigation of flow around a ball valve.

Frobenius et al. [96] presented numerical simulations using three dimensional CFD code Cns3d and experimental studies on cavitating flow through a centrifugal pump impeller. For visual analysis of cavitating flow, they employed a digital video camera used together with a stroboscopic light source. A similar camera is planned for the present work.

2.5 Inferences for current research

The literature review has helped in gaining a better understanding of the problem situation. Labyrinth seals are designed to provide a highly dissipative flow path between high and low pressure regions. The finer geometric variations in the seal configurations can be made possible only with the ability to understand the minute details of flow taking place in the passages of seal cavities. Since one dimensional methods and empirical correlations are virtually inapplicable in labyrinth seal design [21] and the interactions of the numerous geometric parameters are complex and do not lend themselves to analytical methods [75], computer simulation of flow through seals and valves is imperative. The review has adequately implied that analysis of flow through annular seals is fundamental to the study of the same for labyrinth seals.

2.5.1 Finite Element Analysis (FEA)

While finite difference and finite volume based discretisations of the flow domains have been widely reported, finite element codes are not prevalent. Vellando et al. [97] had concluded that as compared to the mixed formulation, the penalty method which is used for solving the Navier-Stokes equations reduces the execution time; however, this is an approximate method that depends on the selection of the penalty parameter which for very small values produces an ill conditioning of the stiffness matrix and for too large values may prevent the system from converging.
The equal order FEA has been shown to provide the correct behaviour of velocity near the walls when compared with the penalty function method which gave oscillations for the velocity component parallel to the wall. The velocity-pressure formulation had been verified for stability of finite element discretizations of Navier-Stokes equations. The mixing length model can be implemented in the calculation of the transport phenomena. Hence, FEA capable of accurately analysing flow through complicated boundaries with the unstructured grid being an inherent feature shall be implemented in the current research.

The finite element equations can be derived from a set of coupled nonlinear Navier-Stokes equations that consist of the conservation of mass and momentums. FEA can use the three-noded triangular element with equal-order interpolations for all the variables of the velocity components, the pressure and the temperature.

Volker and Gunar [98] presented a numerical study of several finite element discretisations to a benchmark problem and found that higher order isoparametric elements are by far the most accurate element for discretising the domain. There are two distinct types of frontal solvers, the first dealing with an assembled matrix and second dealing with finite element codes and an element by element paradigm. While the former is a more general approach that can be applied to assembled matrices arising from any physical application, the second is more popular among finite element codes since it is easily interfaced with finite element data structures. Furthermore, significant savings in memory can be achieved with the latter approach since it does not require assembly of the global matrix.

In the sequential frontal solver for finite element codes, a front sweeps through the mesh, one element at a time, assembling the element stiffness matrices into a frontal matrix. The distinction from the standard assembling procedure is that, as soon as all of the contributions for a given degree of freedom have been accumulated, it is eliminated from the frontal matrix using standard Gaussian elimination. The front then continues to consume elements and eliminate degree of
freedom whenever possible, until the entire domain is transverse and all degree of freedom are eliminated from the linear system. Thus, in the frontal solver the operations of assembling and elimination occur simultaneously and, as a result, the global stiffness matrix never needs to be fully assembled. Of the many techniques available, the direct elimination frontal solution method of Irons [99], has proved to be very effective for solving positive matrices.

For the current research, the foregoing favours the preparation of a numerical model based on classical finite element approach dealt by Taylor and Hughes [100]. This method employs the usage of isoparametric four noded elements for pressure and eight noded elements for velocities. Literature reveals that the flow conditions inside a labyrinth seal would not be fully developed until a decent distance from the inlet. Possibly because of this, the range of Reynolds number applicability of laminar flow solver codes to turbulent flow through annular and labyrinth seals has not been found in the literature, which would be taken up in the present work.

2.5.2 Commercial Computational Fluid Dynamics (CFD) code

Literature reveals that in the computer simulation of labyrinth seals, commercial CFD codes lend appreciable support. The current work would employ the Fluent 6.2 code [22]. This is a structured/unstructured grid solver with an interesting ability to adaptively vary the grid. It solves the time averaged 3D Navier-Stokes equations and represents the turbulence with any one of three turbulence models viz. the classical \(k-\varepsilon\) model, a renormalized version called RNG, and a full Reynolds stress model.

In the standard \(k-\varepsilon\) model of Fluent, Reynolds stresses are modeled using the Boussinesq hypothesis, where the eddy or turbulent viscosity \(\mu_t\) is computed using turbulent kinetic energy \(k\) and its rate of dissipation. The term \(G_k\) is the generation of \(k\) due to the turbulent stress. The model constants are usually kept at their default
values. The RNG model evaluates the effective viscosity by sensitising the constant $C_\mu$. The Reynolds stress model computes the individual Reynolds stresses via a differential transport equation.

In addition to wall boundary conditions, and inlet and outlet velocity/pressure values, additional information has to be provided to *Fluent*, particularly the choice of the turbulence specification method. Since the flow is wall-bounded where the inlet involves a turbulent boundary layer, the Intensity and Length Scale method is an option. The length scale is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows. The solver would compute it upon the specification of the radial clearance in the hydraulic diameter field.

The most common method of specifying $k$ is to use an estimate of the turbulence intensity defined as the ratio of the root-mean-square of the velocity fluctuations to the mean flow velocity. A value of 10% is the frequently assigned value. All these turbulent models are largely valid for turbulent core flows covering the regions somewhat away from the walls.

When the flow to compute involves walls, turbulent flows in the regions close to the walls are affected by the presence of the walls. First, the mean velocity field is affected through the no-slip condition that has to be satisfied at the wall. Turbulence also is changed by the presence of the wall. Very close to the wall, turbulence is dampened, due to the presence of walls. Towards the outer part of the near-wall region, however, turbulence is rapidly augmented by the production of turbulent kinetic energy due to Reynolds stresses and the large gradient of mean velocity. The near-wall modelling significantly impacts the fidelity of numerical solutions, since walls are the main source of mean vorticity and turbulence. It is in the near-wall region that the solution variables change with large gradients and the momentum and other scalar transports occur most vigorously. Therefore, accurate representation of the flow in the near-wall region determines successful predictions of wall-bounded
turbulent flows like the one in a seal.

Numerous experiments have shown that the near-wall region can be largely subdivided into three layers. In the innermost layer, called the viscous sub-layer, the flow is almost laminar-like, and the molecular viscosity plays a dominant role in momentum and heat or mass transfer. In the outer layer, called the fully-turbulent layer, turbulence plays a major role. Finally, there is an interim region between the viscous sub-layer and the fully turbulent layer where the effects of molecular viscosity and turbulence are equally important.

There are two approaches to modelling the near-wall region. In one approach, referred to in the literature as the wall function approach, the viscosity-affected, inner region viz. viscous sub-layer and buffer layer is not resolved. Instead, semi-empirical formulas called wall functions are used to bridge the viscosity-affected region between the wall and the fully-turbulent region. However, those semi-empirical correlations have been established using two-dimensional flat plate experiments results, and do not convey an appropriate model accounting for curvature effect in three dimensional boundary layer as seen in seals or other circumferentially bounded flow. Indeed, in most high-Reynolds-number flows, the wall function approach substantially saves computational resources, because the viscosity-affected near-wall region, in which the solution variables change most rapidly, does not need to be resolved.

The wall function approach is popular, because it is economical, robust, and reasonably accurate. It is a practical option for the near-wall treatments for industrial flow simulations. The wall function approach, however, is inadequate in situations where the Reynolds-number effects are omnipresent in the flow domain in question; consequently, the hypotheses on which the wall functions are based on cease to be valid. Such situations require near-wall models that are valid in the viscosity-affected region and accordingly integrable all the way to the wall. In another approach, which may be called the near-wall modelling approach, the viscosity-affected region is
resolved with a mesh all the way to the wall, including the viscous sub-layer. Fluent provides both the wall function approach and the near-wall modelling approach. In the simulation cited here, the wall function approach is implemented by successively adapting meshes to have $y^+$ values lying between 30 and 500.

**2.5.3 Artificial Neural Network (ANN) and Genetic Algorithm (GA)**

Labyrinth flow can be approached as contraction/expansion taking place through a series of throttling orifices/nozzles. This initiates the development of a semi theoretical-semi experimental model for determining the pressure drop occurring across simple labyrinth seals. Based on this model and a surrogate model, the current research can consider employing ANN and GA techniques respectively in association with CFD analysis for suitably carrying out optimisation studies.

**2.5.4 Investigations on advanced labyrinth seals**

Although much work is found in the literature on the analysis of flow through plain annular and simple labyrinth seals, not much is available on the effect of turbulence flow through labyrinth seals having curved profiles in the grooving pattern/cavity geometry. These three dimensional flows over/through curvatures not necessarily oriented in the streamline direction would be complex enough.

Literature reveals the existence of radial and azimuthal reactions in the advanced seals, the inertial forces also gain prominence besides the viscous effects, leading to better sealing. The current work envisions conducting computer simulation on seals with newer and complex profiles, reducing the development costs.

The presumed application for the current research work demands the employed seal to be least cavitating. No literature is visible on cavitation investigations on seals, which shall be taken up in the current work.

**2.5.5 Cavitation studies on conical valve**

Discharge through conical valves measured through computer simulation is rarely reported in literature which shall be undertaken in the present work. Accelero-
meters/microphones placed either on the downstream valve flange or on the pipe just
downstream of the test valve were tried to measure the intensity of cavitation [94],
but the cavitation noise and vibration were barely detectable over the normal
turbulence flow levels. This prompts innovative application of newer methods for
measuring cavitation intensity like the digital image processing analysis and bacterial
testing, in the current work.

2.6 Development of research methodologies

The need for this research arises because of the inability of the available
seals and conventional conical valve in meeting the high performance targets under
the imposed stiff conditions. Also in most cases, liquid flow through labyrinth seals
and labyrinth-added valve designs are not amenable for analytical treatment. These
aspects imply ample scope for carrying out wide range of research investigations on
the effects of newer labyrinths on seals and conical valve, determination of their
discharge and cavitating flow characteristics and verification of their capability in
meeting the specified demanding requirements.

Usage of flow visualisations and available theoretical models form the initial
stages of the labyrinth seal research methodology. Based on the insights gained, a
couple of semi theoretical-semi experimental formulae involving newer coefficients
are proposed to be developed. Side-by-side application of FEA/CFD based
numerical studies and experimental investigations form the main theme of the
investigations.

Labyrinth seal optimisation methodologies are required to be newer since the
highly coupled and non linear influences of different parameters on the pressure drop
ratio rendered conventional optimisation difficult. With Artificial Neural Network
(ANN), it is easier to incorporate the optimisation inputs. So, an ANN model tool in
association with the semi theoretical-semi experimental formulae to be developed is
devised as the optimisation tool for circular-grooved square and triangular cavity
labyrinth seals.
For the optimisation of helical-grooved square cavity labyrinth seals, CFD and experimental investigations would be made on several such seals. Using the obtained data pool, a pressure drop equation would be obtained empirically using regression analysis. This surrogate equation is mooted for optimising helical-grooved square cavity labyrinth seal geometry by using GA technique. No such previous work employing ANN and GA techniques for labyrinth seal optimisation has been come across in the literature.

Regarding the conical valve, in order to check the comparatively better performance of LCV over that of CCV, the following three different methodologies would be applied:

(i) CFD analysis using *Fluent* with a two-phase mixture model based on the volume of fluid approach

   An unsteady approach based CFD analysis can be employed in realistically modelling the cavitation phenomena. The transient flow field simulation can avoid the occurrence of sharp pressure gradients during the simulation of cavitation. The flow pattern in the valves includes recirculation regions which may have some intrinsic flow unsteadiness. Furthermore, oscillations in pressure and velocity can occur in the flow due to cavitation. Hence, even though the overall flow is steady, for the sake of cavitation simulation, a transient approach is to be necessarily implemented [101].

(ii) Visualisation of cavitating flow using innovative Digital Image Processing (DIP) techniques

   Though cavitation is an evident phenomenon, the nature of the cavitating flow is different in various situations. Arcoumanis et al. [102] investigated into cavitation in the real sized multi hole Diesel nozzles. Flow visualisation in one of the six holes is achieved by incorporating a quartz window into the holed wall and using a fast high resolution CCD camera equipped with high magnification lenses. The present work will devise a suitable method of cavitation visualisation.
Chahine et al. [103] employed an acoustic bubble spectrometer to measure the bubble size distribution in a cavitating flow. A high speed micro video system is also used to take videos of the cavitation bubbles. The current work also contemplates to capture pictures of cavitation bubbles and later indirectly quantify cavitation intensity using DIP techniques [104].

(iii) Novel qualitative estimation of cavitation intensity through a biometric measurement of growth rate of a bacterial colony which is significantly influenced by the intensity of cavitation.

Cavitation enhances chemical reactions, oxidises aqueous compounds and disrupt microorganisms. Gogate [105] had employed cavitation for waste water treatment towards removal of pathogenic organisms. This process sparked another indirect cavitation intensity evaluation technique to be attempted in the present work.

The second and third methodologies devised for the first time in the analysis of cavitation have characteristic high sensitivity to measure the extent of cavitation in a liquid flow. The third technique in particular is a novel application of bio-sensing methodology.

2.7 **Scope of the current research**

Based on the foregoing, the broad scope of the research work is to conduct different investigations on the suitability of various seals and valve geometries towards fulfilling the respective highly demanding preset requirements. The theoretical, numerical and experimental investigations undertaken would have the following agenda:

1. To predict pressure drop taking place in annular seals using available theory, developed FEA code, CFD software analysis and experiments.
2. To conduct flow visualisation studies on different labyrinth geometries and to understand the flow pattern and devise a theoretical model for pressure drop.
3. To predict the pressure drop in labyrinth seals using available empirical models and compare results with FEA predictions, CFD analysis and experiments.
4. To conduct ANN and GA based optimisation studies on circular-grooved and helical-grooved labyrinth seals respectively.

5. To determine the effect of radial clearance, seal length, cavity geometry, flow pattern and turbulence on the pressure drop characteristics of circular, sinusoidal and helical-grooved labyrinth seals and to ultimately determine the seal capable of meeting the targets.

6. To study the discharge and cavitation characteristics of CCV using CFD analysis and experiments.

7. To investigate the effects of labyrinth grooves on the discharge and cavitation characteristics of LCV using CFD analysis and experiments.

8. To estimate the intensity of cavitation in CCV and LCV using Digital Image Processing (DIP) analysis and bacterial testing.

2.8 Overview of the research thesis

The research thesis is organized into seven chapters. The previous Chapter I has broadly introduced the research problems and the present Chapter II presented a detailed literature survey providing more insights, inferences providing discussions on how the research has to be taken forward, research methodologies, scope and this overview. The chapters following the present one are briefly outlined below:

Chapter III starts with visualisation tests to observe the flow pattern in Circular-grooved Square cavity Labyrinth Seals (CSLS). Development of a laminar FEA code is described in analysing water flow through an annular seal and one each configuration of CSLS and Circular-grooved Triangular cavity Labyrinth Seals (CTLS). Later CFD analysis carried out using Fluent code is dealt in predicting the velocity distribution and pressure drop of the annular seal and different configurations of CSLS and CTLS. Numerical and theoretical results are compared with experimental results. This chapter clearly brings out the inadequacy of the annular seals and enables the formulation of a theoretical model for pressure drop prediction.
Chapter IV explains the framing of two different semi theoretical-semi experimental models; one each for CSLS and CTLS, by involving three new parameters named virtual cavity velocity, vortex loss coefficient and cavity loss coefficient. These models are associated with ANN models to identify the optimal configurations of the above two types of labyrinth seals. This chapter concludes that the optimised seals are far from meeting the targeted pressure drop ratio.

Chapter V brings out the development of Circular-grooved Curved cavity Labyrinth Seals (CCLS) using CFD analysis. The predicted flow patterns are observed to be close to observations in visualisation tests. Experimental investigations showed that CCLS are better than both the optimal CSLS and CTLS. CFD analyses explain the changed flow pattern responsible for enhancement of flow resistance. This chapter also deals one Sinusoidal-grooved Triangular cavity Labyrinth Seal (STLS) having slightly better ability than the CCLS. Yet, none of these seals have pressure drop ratio values near to the target. Hence, CFD analysis and experimental investigations on Helical-grooved Square cavity Labyrinth Seals (HSLS) and optimisation of HSLS profile using surrogate model based Genetic Algorithm (GA) are undertaken. It is found that the pressure drop ratio of the optimal HSLS is about 10% below that of the STLS profile. Hence, Helical-grooved Triangular cavity and Curved cavity Labyrinth Seals (HTLS,HCLS) are analysed and tested. This chapter brings out the testing of various geometrical configurations of HCLS eventually leading to one which just goes beyond the targeted pressure drop ratio value. CFD based cavitation analysis of selected seals is also highlighted.

Chapter VI elaborately describes computer simulation and experiments on Conventional Conical Valve (CCV) and Labyrinth Conical Valve (LCV). The CFD analyses indicated that labyrinths reduce cavitation intensity. This chapter goes on to explain the successful implementation DIP and bacterial techniques in indirect quantification of cavitation intensity and verification of less cavitation in LCV.
Chapter VII is the concluding chapter where the major findings of the research work are listed out along with suggestions for future work. A bibliography of the literature referred to for carrying out the research work is included after this chapter followed by the publications arising out of this research work.