Chapter – IV

DESIGN AND DEVELOPMENT OF DYE FLOW CELL FOR SLM

4.1 Introduction

In this chapter several conceptual dye cell geometry designs are discussed. CFD modeling as well as flow visualization studies are reported for selected designs.

Several dye cell geometries such as converging, diverging, straight, trapezoidal, parallelogram, rectangular and cell with convex inner surfaces have been extensively used to achieve high flow velocities in the dye active region [4.1 – 4.7]. The dye cell design has to be such that the flow channel in the dye laser reduces the thermal and pressure variations in the flowing dye solution particularly in the vicinity of the active volume. To some extent the power and repetition rate can be augmented by increasing the flow velocity of the dye solution in the flow channel. Excessive self-heating due to wall friction, disturbances to the streamline, eddies due to the turbulence, local cavitation in the dye solvent are some of the limiting conditions of increasing the flow velocity for a high repetition rate laser. A judicious design of the dye cell flow channel and appropriate selection of active volume will help to solve the problems related to dye flow.

A more serious problem for a SLM dye laser is the instantaneous flow velocity and pump power fluctuations. As the pump pulse energy absorbed by the dye medium degrade the optical quality of the flowing dye, it is imperative that every pump pulse must see a fresh dye volume. The high repetition rate operation demands relatively high flow velocity in the dye cell. A well known consequence of the continuity equation is that, the reduction in the cross section area results in higher flow velocities; increase in velocity is always at the expense of the static pressure. The increase in the flow velocity in a constricted channel causes turbulence to set in the dye solution flow. Such turbulences lead to non-uniformity in the dye refractive index [4.8], resulting in a change in the optical path length of the cavity. It is important to eliminate the turbulence by design and ensure high optical quality laminar flow. The
imbalance between the heating rate and cooling rate of the active volume is primarily responsible for temperature fluctuations in the dye lasers. The heating is caused primarily by the radiation less transitions in the dye molecules, while the cooling is achieved predominantly by heat transfer from the dye solution to the walls, as well as replacement of the volume. It is reported in the literature that high flow velocities in the dye cell can suppress the thermal effects in the dye laser [1.2, 4.1, 4.2, 4.9, 4.10]. High flow velocities may generate local flow induced vibrations which are difficult to eliminate. The flow velocity allows several active volume clearances between pump pulses. Operation at high repetition rate demands a high average flow velocity. The maximum dye flow velocity is limited by cavitations, which destroy the optical homogeneity of the dye medium.

In case of narrow linewidth dye lasers, it is advantageous to utilize the dye cell where its windows are tilted at an angle relative to the optic axis. The wedge type geometry is quite effective in reducing parasitic oscillation in the dye cell. The antireflection coatings are used to diminish the unwanted parasitic oscillations. Roncin and Dammay have designed demountable trapezoidal dye cell to avoid such unnecessary effects [4.5]. Duarte and Piper had used trapezoidal dye cell cross section area of 11 mm x 1 mm with the end window tilted at $\sim 77.7^\circ$ to the long (optic) axis. An alternative to the trapezoidal dye cells parallelogram and rectangular dye cells oriented at an angle to the plane formed by the cavity axis could be used [1.29].

4.2 Fluid Flow

The fluid can flow through a channel as a laminar or a turbulent flow. In a laminar flow there is no lateral mixing, thus all fluid elements keep their position relative to the cross section of the flow channel, the viscosity effects are more dominant than the inertial effects. The fluid in contact with the surface at the boundary is stationary, but all the other layers slide over each other. At higher flow velocities, turbulence appears and vortices form which leads to lateral mixing. In a turbulent flow, the velocity of the fluid particles at a point continuously undergoes change in both magnitude and direction. Between laminar and turbulent flow regimes there is also a transitional regime with turbulence in the center of the region and laminar near the boundaries. The velocity profile between the two flow boundaries of a channel is parabolic in shape for laminar flow. Figure 4.1 shows the velocity profile of laminar and turbulent
flow in a circular pipe. The critical velocity where laminar flow changes to turbulent flow depends on four parameters; the hydraulic diameter of the flow channel, viscosity, density and average linear flow velocity of the fluid. The combination of these four quantities provides the dimensionless Reynolds number given by

\[ Re = \frac{\rho v D}{\mu} \]  

(4.1)

Fig 4.1: Velocity profile of a laminar and a turbulent flow in a pipe

where \( D \) is the hydraulic diameter, \( v \) is the average velocity of the fluid, \( \rho \) is the density of the fluid and \( \mu \) is the viscosity of the fluid. The Reynolds number can be used for identifying the fluid flow velocity either in laminar or turbulent region. In a pipe flow, if the Reynolds number is below 2100, the flow is laminar. The Reynolds number is above 4000 for turbulent flow. The Reynolds number in between 2100 to 4000 is said to be in the transitional zone [1.5, 4.11 – 4.14].

4.2.1 Boundary Layers

A boundary layer is defined as that part of a moving fluid in which the fluid motion is influenced by the presence of a solid boundary due to the viscosity of the fluid and adhesion of fluid at the surface. If the fluid layer does not adhere, the velocity should be free stream velocity \( U_\infty \) everywhere in the flowing fluid. However, most fluids exhibit no slip condition at the surface, that is, the fluid velocity at the solid-fluid interface is zero and the velocities close to the solid surface are small. In the boundary layer, velocity gradients are large enough to produce significant viscous stresses and
dissipation of mechanical energy. Outside the viscous boundary layer, in the free stream, velocity gradients and viscous stresses are negligible. The fluid velocity varies from zero at the wall to a velocity almost equal to that of the free stream (99% of $U_\infty$) [4.15]. In a closed channel, boundary layer extends to the center of the flow channel. At the onset of the turbulence, a turbulent boundary layer is formed, which consists of three zones: the flow in the boundary layer very near the surface, the viscous sub layer is essentially laminar; the zone between fully developed turbulence in the turbulence zone and the region of viscous sub layer is a transition zone called the buffer layer. For flow in a pipe or duct the boundary layers will eventually meet at the center and at this point they cannot grow further. The velocity profile reaches an asymptotic state where the flow is said to be fully developed. The effects of viscous friction are then felt over the whole cross section of the tube. The length of the entrance region of the pipe necessary for the fully developed flow to be established, is called the transition length [4.13 – 4.15].

The boundary layer thickness for laminar and turbulent boundary layer can be approximately estimated [4.13 – 4.15].

\[
\frac{\delta}{x} \sim \frac{5}{\sqrt{Re_x}} \tag{4.2}
\]

\[
\frac{\delta}{x} \sim \frac{0.16}{(Re_x)^{1/7}} \tag{4.3}
\]

**Flow Induced Vibration**

The fluid induced vibration can originate from a number of system conditions such as

- High turbulence
- Pressure pulsation
- Cavitation

The high turbulence can excite the vibrations. An inherent feature of the turbulent flow is a random pulsation of pressure. This process generates a broad spectrum of frequencies, which rise with increasing fluid velocity. The formation and shedding of eddies in places where fluid is forced to change flow direction abruptly causes additional pressure pulsations. The broad spectrum of frequencies generated by a turbulent flow increases the risk of resonance.
Pressure pulsation originates from the operation of fluid handling equipment like pumps. In the gear pump, the pressure of the liquid delivered by the pump pulsates at the frequency at which teeth of the gear pump pass through the pump discharge. Thus, this frequency depends on the number of gear teeth and rotational speed (RPM). If one of the natural frequencies of the components in the flow system is close to a frequency of the flow induced vibration, resonance may increase vibration amplitudes to an unacceptable level.

**4.2.2 Cavitation**

Cavitation is a term to describe the process, which includes nucleation, growth and implosion of vapour filled cavities and is important in fluids, which have significant vapour pressure. Cavitation takes place in the dye cell when the static pressure of the liquid falls below the vapor pressure of the liquid at a given temperature. When the pressure in the liquid is reduced vapor bubbles appear. The vapour bubbles grow till low pressure occurs. Later, the pressure in the flow channel rises due to decrease in average velocity; the cavities repeatedly collapse leading to local high pressure and cause sonic waves. The degree of cavitation can be determined by a dimensionless parameter typically known as cavitation number \( K \)

\[
K = 2 \frac{(P_d - P_v)}{\rho v^2}
\]  

(4.4)

where \( P_d \) is the pressure, \( P_v \) vapor pressure, \( \rho \) density and \( v \) bulk velocity.

The numerator in the above equation corresponds to the static pressure, which resists the cavitations while the denominator corresponds to the dynamic pressure, which promotes cavitations. Hence the reduction in the local pressure inside the dye cell should be limited to the vapor pressure of the solvent. Because of sudden changes in flow conditions, local heating and lower surface energy can also cause the generation of cavitation (vapour bubble) in the flow. The cavitations result in vibrations as well as noise, which cause short term frequency fluctuations [4.15]. The formation of vapour bubbles (cavitation) contributes to damage of the dye cell windows in several ways.

- The bubble harshly affects the heat transfer at the solid-liquid interface from the quartz window to flowing dye solvent.
The bubble degrades the optical quality of the lasing cavity and increases the possibility of scattering for pumping and lasing radiations.

The flow geometry, the operating pressure and the pressure drop, the solvent properties are chosen judiciously to eliminate cavitation.

### 4.3 Flow Requirement for SLM Dye Laser

Ideally, the liquid exposed to the pump beam should be replenished in the dye cell before the arrival of the next pulse, so that the gain medium is fresh. It is important to minimize thermal induced refractive index gradient in the dye cell to achieve high beam quality of the dye laser. The pump beam energy absorbed in the dye medium, at least one fourth of the energy results in heating of the dye solvent. The temperature gradient due to heating produces a refractive index gradient \( \frac{dn}{dt} \), which results in a deflection of the dye laser beam. The value of \( \frac{dn}{dt} \) depends on the thermo optical properties of the solvent. The pump beam does not get absorbed uniformly throughout the dye cell due to Beer Lambert exponential law. The cell width and dye concentration determine the energy deposition in the dye solvent. The flow rate is sufficient to flush out the optically pumped dye solution volume after each pump pulse. The velocity profile again is not uniform in the dye cell. A boundary layer of very slow moving dye solution exists at the window surface. The velocity must be sufficiently high to remove the slow moving layer, where the laser beam intensity would be the maximum. Thus,

\[
v_g > \frac{V}{T_1}
\]  

(4.5)

where \( v_g \) is the average volume flow rate, \( V \) is the cell volume being optically pumped and \( T_1 \) is the repetition period. The number of cell volumes replaced per pump pulse is recommended to be two or more for eliminating the thermal effects.

Optical damage to the dye cell windows occurs in most dye lasers at some point because the intensities of the lasing beam and pump beam intensities are generally very close to the damage threshold of the dye cell windows. The damage threshold depends on window material, surface quality, cleanliness of the window, heat transfer, flow conditions and laser power density. In practice, for a given pump power density
liquid-solid interface suffer higher damage at much lower power densities than vapor solid interfaces. One of the important causes of damage at lower power densities is the deposition of photo degraded products on the dye cell windows. The dye cell windows are not merely passive surfaces that collect the photo degraded products, but they also actively participate in the photochemical reactions. The thermal conductivity of the dye cell windows may also play an important role in the damage mechanisms by affecting the local heat transfer. The normally used quartz has less thermal conductivity than magnesium fluoride. It has been observed that the damage threshold of the quartz windows shows major improvement, when the solvent used is Dioxane. However, it should be remembered that the photochemical stability is generally poor under higher power densities.

To avoid extensive time consuming experimental studies it was decided to study the dye cell design suitable for high repetition rate SLM pulsed dye laser, through computational modeling, in a general purpose computational fluid dynamics (CFD) software. The rectangular dye cell geometry was chosen for the study of single mode dye laser.

### 4.4 Computational Fluid Dynamics (CFD)

CFD is a powerful tool used for the prediction and analysis of fluid flow. It is able to offer three dimensional, time dependent and fairly accurate solutions to the highly coupled differential equations that govern fluid flow in a domain and help conduct computational experiments. Thus a large number of designs can be tested rapidly without spending any time or money for tedious experimental investigations.

#### 4.4.1 Overview of CFD Study and the Software

The main aspects of numerical modeling performed within the scope of this thesis are presented here for a general physical and mathematical understanding of the simulations used. The CFD code contains discretisation techniques suitable for the treatment of the key transport phenomena, convection and diffusion as well as the source terms and the rate of change with respect to time. The underlying phenomena are complex and non-linear, so an iterative solution approach is required. Following steps are taken to use the CFD package to carry out the analysis on hand [4.16].

97
A control space is defined where the equations are to be solved. A geometrical model is built using Solid Works [4.16], pre-processing the model to generate the grid; each grid point is given an initial value chosen by the user. The grid generation for the dye cell flow domain was carried out with ANSYS ICEM CFD and ANSYS CFX [4.16]. One of the important factors when it comes to obtaining convergence when performing CFD simulations is the use of good quality mesh. Quality of hexahedral mesh in ICEM is calculated as the relative determinant of each cell. The relative determinant is the ratio of the smallest determinant of the Jacobian matrix divided by the largest. A determinant value of one would indicate a perfect regular mesh element; zero would indicate one or more degenerate edges and minus one would indicate completely inverted elements. The other aspects that should be considered for the quality of the mesh are aspect ratio, the skewness and the internal angles for the element. The better this guessed starting point is the faster a solution is reached.

Boundary conditions are specified for the flow domains at the walls, inlets, outlets and openings. These are known values of velocity, mass flow, heat load etc, at the inlet, outlet and the walls.

Next is to set the time steps and selecting numerical sub-models. The domain is split up into a number of cells (control volumes around a grid point) known as meshing. The mesh does not necessarily have to have the same density in the entire domain. The equations are discretised in the centre of the computational cells. This means that the partial differentials are translated into equations with coefficients that form a matrix. The algorithm in the solver solves this matrix.

*Monitoring the iterative solution process*

For each cell of the mesh the governing equations of mass, momentum and energy are solved together with the model related equations, before the computation moves on to the next cell. When the calculation is finished for all the cells, one iteration step is finished. The results are compared to the solution of the previous step. If the difference between the two solutions is larger than the specified accuracy, calculation for all cells is repeated, starting from the last solution as the start condition.

*Analyzing the solution obtained*

The results can be organized, displayed and analyzed by post processing tools. Most of them work with graphical user interface.
The main elements of the CFD code are shown in figure 4.2. It contains three main elements.

i. A pre-processor
ii. A Solver
iii. A Post Processor

4.4.2 Pre-processor

The pre-processor consists of the flow problem to a CFD program by means of an operator friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The steps involved in the pre-processor stages are definition of flow geometry or computational domain, grid generation, selection of physical and chemical phenomena, defining fluid properties and specifying boundary conditions [4.15].
4.4.3 Solver

ANSYS CFX is a finite volume method code. The approach involves discretising the flow domain into finite control volumes using ANSYS ICEM CFD mesh.

4.4.4 Selection of Physical Sub Model

The fluid flow equations solved in a general purpose CFD code are not the exact governing Navier Stokes equations. This is because the exact equations either cannot be solved for physical flows of interest as is the case with turbulent flow or because the exact governing equations are not completely known. Hence simplified physical sub models are employed. They consist of a set of approximate equations in part derived from empiricism and physical reasoning. One has to select the proper sub model for turbulence to be implemented in the simulation.

4.5 Turbulence Models

It is very crucial that the user must have a good understanding of the physical mechanism of some of the turbulence sub-model as well as good knowledge of the limitations of the sub model. The turbulent flow contains a wide range of length and time scales. For fluid flow, the scales are dependent on characteristics of flow and the flow dimensions. Several approaches for turbulence modeling have been proposed. They are broadly classified into three categories.

*Direct Numerical Simulation (DNS)*

All the turbulent motions are resolved by solving directly the Navier Stokes equations that govern fluid flow without additional modeling. This approach requires a very large number of grid cells and thus requires large computational resources for practical applications.

*Large Eddy Simulation (LES)*

Although, this approach allows employing fewer grid cells than DNS but still impractical for complex flow geometries. In this model, the smallest turbulent motion is resolved by Navier Stokes equations. The finest eddies are either ignored or modeled.
Reynolds Averaged Navier Stokes Model (RANS)

The Navier Stokes equations are time averaged and thus the equations so obtained do not aim to resolve the turbulent motions, but to provide the time averaged characteristics of flow quantities. Since, the RANS model is not trying to resolve the turbulent motions, the grid needs only to be fine enough to capture the important time average feature of the fluid flow.

4.6 The Two Equation Model

Two equation models are the most commonly used models for turbulence and engineering problems. They introduce two new transport equations that represent the turbulent properties of a flow, like convection and diffusion of turbulent energy. The most common transport variables are the turbulent kinetic energy and either the turbulent dissipation or the turbulent specific dissipation. The turbulence dissipation determines the length or time scales of the turbulence. The velocity scale is often modeled using an extra transport equation for the turbulent kinetic energy, given by

\[ k_1 = \frac{1}{2} u' \]  \hspace{1cm} (4.6)

where \( u' \) represent the turbulent velocity fluctuations. The length scale is calculated as a function of the turbulent kinetic energy dissipation (\( \epsilon \)) or the turbulent kinetic frequency (\( \omega \)).

4.6.1 The k-\( \epsilon \) Model

The k-\( \epsilon \) model assumes that the eddy viscosity is related to the turbulent kinetic energy and turbulence dissipation rate according to relation

\[ \mu_t = C_{\mu} \rho \frac{k^3}{\epsilon} \] \hspace{1cm} (4.7)

The values of the turbulent kinetic energy and the eddy dissipation come directly from their differential transport equations respectively

\[ \frac{\rho D_k}{Dt} = \nabla \left[ (\mu + \frac{\mu_t}{\sigma_k}) \nabla k \right] + P_k - \rho \epsilon \] \hspace{1cm} (4.8)

\[ \frac{\rho D\epsilon}{Dt} = \nabla \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \nabla \epsilon \right] + \frac{\epsilon}{k} \left( C_{\epsilon_1} P_k - C_{\epsilon_2} \rho \epsilon \right) \] \hspace{1cm} (4.9)
where $C_{\epsilon 1}$, $C_{\epsilon 2}$, $\sigma_k$ and $\sigma_\epsilon$ are constants $P_k$ is the turbulence production due to viscous and buoyancy forces.

The main drawback of this model is a poor prediction of the flow behavior in the case of non-equilibrium boundary layers. The reattachment point during separation flow calculation is usually under predicted. Errors occur in the magnitude of the local heat transfer and as a consequence, the overall device performances are solved incorrectly [4.14, 4.15, 4.17].

4.6.2 The k-ω Model

The k-ω model was developed in order to improve the predictions in the near wall region and reduce the errors in adverse pressure gradient calculations. In order to define the turbulent eddy viscosity, the k-ω model uses a frequency scale ($\omega$) called also specific turbulent dissipation rate.

$$\mu_t = \rho \frac{k_1}{\omega}$$  \hspace{1cm} (4.10)

In this model instead of eddy dissipation a turbulent eddy frequency ($\omega$), which is related to the length scale, is used for the formulation of equations.

$$l = \frac{k_1^{1/2}}{\omega}$$  \hspace{1cm} (4.11)

instead of the formulation used for the k-ε model

$$l = \frac{k_1^{3/2}}{\epsilon}$$  \hspace{1cm} (4.12)

So the turbulent kinetic energy equation is written

$$\rho \frac{Dk}{Dt} = \nabla \left[ \left( \mu + \frac{\mu_\epsilon}{\sigma_k} \right) \nabla k_1 \right] + P_k - \beta' \rho k_1 \omega$$  \hspace{1cm} (4.13)

And the equation for the eddy dissipation rate $\omega$ is written

$$\frac{D\omega}{Dt} = \nabla \left[ \left( \mu + \frac{\mu_\epsilon}{\sigma_\omega} \right) \nabla \omega \right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2$$  \hspace{1cm} (4.14)

The major advantage of the k-ω model is the robust and simple way the near wall region is handled. Contrary to the k-ε model the k-ω model does not involve complex non-linear damping functions to take into account the near wall low Reynolds effects. The main weakness of the k-ω model is the strong sensitivity of the solution to the free stream $\omega$ values [4.14, 4.15].
4.6.3 The Shear Stress Turbulence Model

In order to get the best from the k – ε and the k – ω models, a new blended model called Shear Stress Turbulence (SST) model is available in the ANSYS CFX. The SST model calculates the flow in the near wall region using a k – ω formulation whereas in the bulk flow the high Reynolds k – ε formulation is employed. A smooth transition between the two formulations is ensured by the use of additional blending factors, which are functions of the wall distance. The SST model has good merits for accounting for adverse pressure gradients and separating flow, but tends to produce too large turbulence levels in the stagnation region and regions with strong acceleration. To make the switch between the models, the equations are multiplied with blending functions and the final result for kinetic energy and the eddy dissipation is

\[
\rho \frac{Dk}{Dt} = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_k^{3.3}} \right) \nabla k \right] + P_k - \beta' \rho_k \omega \quad (4.15)
\]

and

\[
\frac{D\omega}{Dt} = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega^{3.3}} \right) \nabla \omega \right] + (1 - F_1) 2\rho \frac{1}{\sigma_\omega^{2.0}} \nabla \omega \nabla \omega + \alpha_3 \frac{\omega}{k} P_k - \beta_3 \rho \omega^2 \quad (4.16)
\]

As in the k – ω model, the CFX implementation of the SST model permits the automatic shifting from the low Reynolds number formulation to the wall function scheme according to the grid resolution. It is obvious that special attention has to be paid to modeling the near wall region. Near the wall, turbulence is dampened and the velocity decreases to zero as viscous forces start to influence the flow. The SST model is used in this thesis for evaluating the dye cell geometries for single mode operation. [4.14,4.15,4.18, 4.19].

4.7 The Finite Volume Method

ANSYS CFX is a finite volume method – FVM code. The finite volume method was originally developed as special finite difference formulations. Each mesh element is translated into a bonded assembly of finite volumes, each of which has its unique form and address and enclose its appropriate node as shown in figure 4.3. In the finite volume method, the volume integrals in a partial differential form of the governing (momentum, energy, mass) equations that contain divergence terms are converted to
surface integrals, using the divergence theorem. These terms are then evaluated as fluxes at the surfaces of each finite volume while ensuring the principle of conservation [4.20, 4.21]. For example, the integral form of the continuity equation for steady incompressible flow is reduced to

\[ \int_S \vec{V} \cdot \vec{n} \, dS = 0 \]  \hspace{1cm} (4.17)

The integration over the surface S of the control volume and n is the outward normal to the surface. Physically, this equation means that the net volume flow into the control volume is zero. Let us consider a rectangular cell

The velocity at the face I is taken to be \( \vec{V}_i = u_i \hat{i} + v_i \hat{j} \). Applying the mass conservation equation (4.17) to the control volume defined by the given cell

\[ -u_1 \Delta y - v_2 \Delta y + u_3 \Delta y + v_4 \Delta y = 0 \]  \hspace{1cm} (4.18)

This is the discrete form of the continuity equation for the cell. Hence the cell values at the cell centers are stored. The face values \( u_1, v_2, \) etc are obtained by suitably interpolating the cell centre values for adjacent cells. Similarly, one can obtain discrete equations for the conservation of momentum and energy for the cell.

Fig 4.3: Control volume defined by a cell
4.8 Conjugate Heat Transfer

ANSYS CFX allows for incorporation of solid domains and modeling of the liquid–solid interaction within one simulation model. In a solid domain the equation for the conservation of energy is simplified since there is no flow within or in and out of a solid and conduction is essentially the only mode of heat transfer. Fluid and solid domains are coupled in ANSYS CFX via conservative domain interface allowing for momentum and energy transfer across the boundaries [4.22]. The conjugate heat transfer will be presented in this thesis for absorption of pump beam in the dye solution. In which the pump beam energy is deposited in a cylindrical volume. This heat source is treated as exponentially decaying volumetric heat source and will be discussed in detail in chapter – VI of this thesis.

4.9 Convergence Criteria

As the number of grid points increases, the error in the numerical solution would decrease and the agreement between the numerical and exact solutions would get better. When the numerical solutions obtained on different grids agree to within the specified level of tolerance specified, they are referred to as grid converged solutions; convergence becomes independent of the grid as the cell size is reduced. Further, refinement of the grid is carried out for a better agreement between the numerical solution and the exact solution. As the guessed value of \( u_g \) tends to \( u \), the linearization and matrix inversion errors tend to zero. The residual \( R_1 \) is the RMS value of the difference between the \( u \) and \( u_g \) on the grid, given by

\[
R_1 = \sqrt{\frac{\sum_{i=1}^{N} (u_i - u_{g})^2}{N_1}}
\]  

(4.18)

It is useful to scale this residual with the average value of \( u \) in the domain. In a good simulation, both iterative convergence error and truncation error would be of comparable level and less than the tolerance level chosen.

4.10 The Dye Cell Geometries and Flow Analysis

The restriction in flow area by the convex-convex, convex-plane, or plane-plane surfaces increases the flow velocity in the region of active volume [2.29, 4.3 – 4.7,
As an alternative to the experimental methods, detailed CFD studies were undertaken for several rectangular dye flow channels of cross sectional area ranging from 5 x 1 mm$^2$ to 20 x 1 mm$^2$ and cell lengths from 30 mm to 80 mm. The entrance to flow channel was either rectangular or tubular. The fluid entering in a flow channel requires a certain length to develop the velocity profile in the flow channel. This length is known as entry length, depends on the type of flow channel, flow velocity and solvent properties. The entrance of the flow channel was rounded to avoid an initial disturbance to the entering flow stream. A FVM with ANSYS CFX solver of general purpose CFD software was used to visualize the flow velocity vectors in the flow domain. The ANSYS CFX – 12 solved the Navier – Stokes equation for the flow domain, which was uniformly, meshed using ICEM CFD mesh. The mesh element consists of tetrahedral cells and prismatic cells near solid-liquid interface. The SST model was used to solve the k-ω model at the wall and solve k-ε model in the bulk flow, where $k$ is the turbulent kinetic energy, $\varepsilon$ turbulent eddy dissipation and $\omega$ turbulent frequency. A blending function ensures a smooth transition between the two models. The quality of meshing has a significant effect on the accuracy of results [4.28 – 4.31]. Theoretically, higher number of mesh elements in the flow geometry provide better accuracy of the results, but the computational time is higher. The grid independent results were obtained by refining the mesh element of flow domain so that with further refinement of the grid the computational result does not change significantly. The grid size near the wall was $\sim 25 \mu m$ and in the bulk $\sim 250 \mu m$. The Intel Pentium Quad Core, 2.8 GHz, 4 GB RAM and 1 TB HDD was used for all runs. A typical solid works flow domain is shown in figure 4.4. The typical mesh for SLM dye laser shell is shown in figure 4.5.

It was observed that the solution obtained from ANSYS ICEM CFD mesh has better convergence than the mesh generated by ANSYS CFX mesh generator. The typical meshes generated by ANSYS CFX and ANSYS ICEM CFD mesh generator are shown in fig 4.6 and fig 4.7 respectively. The convergence time for ANSYS ICEM CFD mesh has almost 50% less than the convergence time for ANSYS CFX mesh for the same geometry, mesh element size and boundary conditions.
Fig 4.4: Typical SLM dye cell flow geometry with tubular entry.

Fig 4.5: Typical unstructured mesh for SLM dye cell with mesh element size of 175 µm.
Fig 4.6: Grid generated by ANSYS CFX

Fig 4.7: The grid generated by ANSYS ICEM CFD
4.11 Inputs and Boundary Conditions

The following boundary conditions were used for these simulation studies: inlet flow velocity 2 m/s, outlet pressure of 1 atm and no slip condition at the solid liquid interface. The no slip condition defines the flow velocity at the solid surface as zero. Here, it is also assumed that at the surface the fluid temperature will be equal to the temperature of the surface. The liquid properties used for simulation were molar weight, density, viscosity, thermal conductivity, specific heat, etc. The simulations were carried out for four different types of dye solvents employed in the calculation.

The flow properties (viscosity and density) of the different solvent are listed in the following table 4.1.

<table>
<thead>
<tr>
<th>Solvent</th>
<th>Ethanol</th>
<th>Glycerol</th>
<th>Ethylene Glycol</th>
<th>Binary Solvent Ethanol and Glycerol (50:50)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Viscosity</td>
<td>1.087 cP</td>
<td>1200 cP</td>
<td>20 cP</td>
<td>200 cP</td>
</tr>
<tr>
<td>Density</td>
<td>789 kg/m³</td>
<td>1261 kg/m³</td>
<td>1115 kg/cm²</td>
<td>1023 kg/m²</td>
</tr>
</tbody>
</table>

Table 4.1: Solvent Properties Used for Computational Fluid Dynamics Analysis

4.12 Dye Cell Geometries and Flow Simulation

Several dye cell geometries have been analyzed and studied with their velocity vector visualization. Some of the typical cell geometries and their velocity vectors are presented in the table 4.2. In all the cases, the flow medium is ethanol. The referred design numbers correspond to those indicated in the table.

Design – I

In this cell of uniform cross section of 19 mm x 1 mm and length of 70 mm, the inlet and outlet headers are circular channel of 8 mm diameter. The simulation results show flow circulation (vortex) near the entrance of the dye cell whose length is approximately one fourth of the cell. The pressure drop across the cell was computed to 101.314 kPa at 1 m/s inlet flow velocity.

Design – II

For this design the flow cross section area was reduced from 19 mm² to 10 mm² (10 mm x 1 mm) and the flow header diameter was reduced from 8 mm to 4 mm. It was observed that flow circulation up to 20 mm and 25 mm length from the inlet was obtained for flow velocities of 2 m/s and 4 m/s respectively. The pressure drop across
the flow cell increased from 112.756 kPa to 146.957 kPa with an increase in the flow velocity from 2 m/s to 4 m/s; the increase in the pressure drop was nearly 30%, indicating the major pressure loss in the secondary flow.

**Design – III**

In design III, the flow length was increased to 80 mm for the cross sectional area of 10 mm$^2$. For this design flow circulation near the entry of the flow cell was seen. The flow circulation extended up to almost 40% of the flow length for inlet flow velocity of 2 m/s. On increasing inlet flow velocity the flow circulation length was increased. The pressure drop across the cell was nearly 116.55 kPa.

**Design – IV**

In design IV, the flow cross section area was reduced to 5 mm$^2$ (5 mm x 1 mm) from 10 mm$^2$ and the flow length was reduced to 65 mm from 80 mm. For this design, there was no flow circulation observed. On reduction of cross sectional area the flow circulation was eliminated. The pressure drop across the dye cell was computed to be 122.964 kPa at an inlet flow velocity of 1 m/s.

**Design – V**

In design V, the change from the preceding design was that the inlet flow header tube diameter was increased to 8 mm from 4 mm and the flow channel length was varied from 70 mm to 30 mm in steps of 10 mm. The flow velocity inside the flow channel was the same for all the dye cells of different flow channel lengths while the pressure drop increased with increasing flow channel length for the same inlet flow velocity. The pressure drop across the cell was 138.604 kPa for 30 mm cell length and it was 152.708 kPa for 70 mm cell length. There was no vortex present in the flow cell for slit entry as well as tubular entry to this flow cell.

**Design – VI**

For Design VI, the flow channel length and the flow cross section area were kept the same as in design V. The flow entry to the cell was smoothened with the introduction of 2 mm radius fillet. The maximum flow velocity components were 2.24 m/s and 11.71 m/s for inlet flow velocities of 0.198 m/s and 0.994 m/s respectively. No flow circulation was observed for all the five simulations carried out for different inlet flow velocities. The pressure drop across the flow cells was 104.04 kPa and 162.773 kPa for 0.198 m/s and 0.994 m/s inlet flow velocities respectively.
Design – VII
For Design VII, converging, straight and diverging sections were introduced in the flow geometry and the flow width was kept 19 mm. The inlet header tube diameter was 8 mm. The flow circulation near the inlet as well as at the outlet was observed. The flow circulation starts immediately after the diverging section of the flow. The pressure drop across the cell was 102.625 kPa at an inlet flow velocity of 1 m/s. The pressure drop across the flow cell was relatively small due to converging and diverging flow sections. Due to convergence and divergence geometry, higher flow velocity was achieved with a smaller pressure drop.

Design – VIII
For Design VIII, the flow channel width was reduced to 5 mm$^2$ (5 mm x 1 mm) from 19 mm$^2$ (19 mm x 1 mm) and flow channel length was 90 mm. This design had also converging, straight and diverging sections in the flow domain. The velocity vectors show flow circulation both near inlet as well as at the outlet of the flow cell. The maximum flow velocity was in the straight region of the flow cell. The pressure drop across the flow cell was 108.118 kPa at an inlet flow velocity of nearly 2 m/s. For converging and diverging flow channels the pressure head loss is minimized due to recovery of pressure in the diverging section.

Design – IX
For this design IX, triangular converging and diverging sections were introduced; the diverging angle was kept 30$^\circ$ for both diverging and converging angles. Total flow domain length was kept 25mm in which 10 mm length was kept straight. The inlet and outlet header areas were also kept 25 mm$^2$. A flow circulation in the outlet was observed at an inlet flow velocity of 1 m/s. The maximum velocity of 4.43 m/s was obtained in the straight section of the flow domain.

Design – X
For this design X, the triangular geometry was changed to a smooth radius of 30 mm with 30$^\circ$ convergence angle and 30$^\circ$ divergence angle. The other dimensions remained the same as in the design IX. Flow circulation at the outlet of the flow channel was observed and the flow velocity was the same as in design IX as the flow cross section dimensions remained unchanged.
Design – XI
For design XI, the converging and diverging channel radii were reduced to 15 mm from 30 mm and keeping all the other dimensions the same. In this design also the flow circulation at the outlet of the flow cell was present while the flow velocity was the same as in designs IX and X.

Design – XII
In design XII, the converging and diverging radius was changed to 151 mm and straight length was kept 10 mm. The flow channel gap was reduced to 0.5 mm from 1 mm unlike in all the other designs; the flow channel length was 124 mm with inlet and an outlet header area of 200 mm$^2$. Flow circulation was observed immediately after straight section at the outlet of the flow cell. The pressure drop across the flow cell was only 103.83 kPa despite relatively higher flow velocities of 19.5 m/s in the straight section.

Design – XIII
In design XIII, a smooth flow entry with 2 mm radius fillet was introduced before the straight section of the flow domain, followed by a diverging section with a divergence angle of 14$^\circ$. There was a flow circulation at the outlet of the flow channel as seen in the earlier designs. The pressure drop across the flow channel was computed to be 106.645 kPa at a flow velocity of 5.32 m/s in the straight section of the flow channel.

Design – XIV
For design XIV, the fillet radius was increased to 3 mm, straight length reduced to half and diverging angle also reduced to 6.73$^\circ$ with flow channel length of 55 mm to keep the inlet and the outlet header area the same. The entry and exit in the cell were with a tube of 4 mm diameter. The maximum flow velocity component was 1.42 m/s for inlet flow velocity of 1 m/s. No flow circulation was observed in this flow geometry. The pressure drop across the flow cell was 101.95 kPa at an inlet flow velocity of 1 m/s.

The converging and diverging flow geometries are useful for high flow velocities in the flow channel as the pressure drops are smaller in higher flow velocities. These types of flow cell geometries can be used for high repetition rate lasers where the flow velocity requirements are high without significant pressure head loss.
Table 4.2: Design Number – I

Flow channel gap 1 mm, width 19 mm and length 70 mm, Inlet outlet tube diameter 8 mm. Flow channel area 19 mm² with slit entry from nearly 50 mm² inlet header.
Table 4.3: Design Number – II

The flow channel cross section area was 10 mm\(^2\) with flow channel length of 76 mm. The inlet tube diameter was kept 4 mm to match the flow channel area. The entry to the flow channel was kept slit area of 10 x 1 mm\(^2\).
Table 4.4: Design Number – III

For this design the flow channel area was also kept $10 \text{ mm}^2$ with flow channel length of 80 mm, inlet as well as the outlet tube diameter was 4 mm with slit entry.
Table 4.5: Design Number – IV

For this design the flow channel area was reduced to 5 mm² with flow channel length of 65 mm. The entry to the flow channel was kept from tube of 4 mm diameter with a slit entry.

<table>
<thead>
<tr>
<th>Flow Domain</th>
<th>Velocity Vectors</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
</tr>
</tbody>
</table>

**Velocity (Vector 1)**
- 5.027e+000
- 3.770e+000
- 2.513e+000
- 1.257e+000
- 0.000e+000

[Velocity vectors in m/s^1]
Table 4.6: Design Number – V

For this design the flow channel area of 5 mm$^2$, 60 mm tube length 8 mm tube diameter with slit entry and exit. The flow channel lengths were varied from 70 mm to 30 mm in steps of 10 mm for different simulations.
Table 4.7: Design Number – VI

The flow channel length was kept 30 mm, flow cell cross section area was kept 5 mm$^2$ and the entry to the cell was made by slitting 8 mm diameter tube. The flow entry to the cell was smoothened with the introduction of 2 mm fillet to avoid the initiation of the turbulence in the flow channel. The inlet flow velocity was altered from 0.198 m/s to 0.994 m/s to obtain the required flow velocities in the flow channel.
Table 4.8: Design Number – VII

Flow channel gap of 1 mm, 19 mm flow width and total flow channel length was kept ~ 75 mm with 5 mm straight section. Inlet tube diameter was 8 mm and 64 mm tube length with converging-diverging flow geometry.
Table 4.9: Design Number – VIII

For this design the flow channel gap was kept 1 mm, flow channel area of 5 mm², flow channel length of 90 mm with 5 mm straight length having converging-diverging flow channels. The flow entry to the dye flow cell was tubular with tube diameter of 5 mm.
Table 4.10: Design Number – IX

This flow channel consists of three sections triangular; converging, straight and triangular diverging. The diverging angle was kept nearly 30° for both converging and diverging sections. The flow cell width was kept 5 mm, flow length 10 mm, flow channel gap 1 mm and both inlet and outlet out areas is 25 mm².
Table 4.11: Design Number – X

This flow cell had converging, diverging and straight sections. The converging and diverging section has 30 mm radius with 30° angle. The inlet and outlet areas were the same ~ 25 mm². The flow cell gap was kept 1 mm and flow cell length 25 mm.
Table 4.12: Design Number – XI

This flow cell has three sections: Converging, straight and diverging, the converging and diverging section radii were 15 mm with inlet and outlet flow areas of 25 mm². The straight channel flow area was 5 mm², with a flow channel length of 25 mm.
Table 4.13: Design Number – XII

This flow cell design has converging, diverging radius of 151 mm and a straight section of 10 mm length with inlet and outlet areas of nearly 200 mm². The straight channel had an area of 8 mm² with straight length 10 mm, flow channel gap 0.5 mm and total flow channel length of 124 mm.
Table 4.14: Design Number – XII

For this design the flow channel gap of 1 mm in the straight section, flows entry with 25 mm² flow areas having 2 mm fillet radius had been analyzed. The total flow channel length of 40 mm, straight section of 12 mm and the divergence angle of 14º.
Table 4.15: Design Number – XIV

The inlet has a fillet of 3 mm radius; flow channel gap of 1 mm for straight length of 6 mm, diverging angle of 6.73°, flow channel length of 55 mm. The flow cell inlet area was 35 mm² and the flow entry was 4 mm diameter tube.
4.13 Selection of Dye Cell for Detailed Analysis

Of the several designs investigated two dye cells with different flow cross section and different flow entries were chosen for fabrication, mainly because of the simplicity in fabrication. These were simulated in detail by the CFD model. The velocity vectors obtained for these two dye cells of dimensions 10 mm x 1 mm x 70 mm (Length, width and height) and 5 mm x 1 mm x 70 mm have been presented in figure 4.8 a, b and figure 4.9 a, b respectively. The velocity vectors for 1 x 10 mm² cell in figure 4.8 shows a flow circulation (~ vortex) near the entrance of the dye cell, which is spread over two third length of the dye cell with inlet flow velocity of 2 m/s and ethanol as solvent. This dye cell has a tubular entry with an inlet tube diameter of 4 mm. The circulation length reduced to half the cell length with a slit entry (1 x 10 mm) to the dye cell (1x 10 mm²) for the same boundary conditions. The mean flow velocity in the flow channel was computed to nearly 2.5 m/s. The circle marked on the dye cell shows the position of active volume for generating the SLM dye laser.

![Velocity vector diagram for dye cell](image)

Fig 4.8 a: Flow velocity vectors for ethanol solvent with 2 m/s inlet flow velocity of 10 x 1 x 70 mm dye cell with tubular entry.

When the flow channel cross section area was reduced from 10 mm² to 5 mm² and tubular entry changed to slit entry, as can be seen in figure 4.9 a; no flow circulation was observed in the velocity vector diagram for the same boundary condition and the
same solvent. No flow circulation was observed in the computational results with tubular entry of 4 mm ID in this dye cell (1 x 5 mm²) as shown in figures 4.9 b.

Fig 4.8 b: Flow velocity vectors for ethanol solvent with 2 m/s inlet flow velocity for 10 x 1 x 70 mm dye cell with slit entry.

Fig 4.9 a: Flow velocity vectors for ethanol solvent with 2 m/s inlet flow velocity for 5 x 1 x 70 mm dye cell with slit entry.
Fig 4.9 b: Flow velocity vectors for ethanol solvent with 2 m/s inlet flow velocity for 5 x 1 x 70 mm dye cell with tubular entry.

The use of solvents with higher viscosity such as binary solvent (50:50 ethanol and glycerol) and pure glycerol allowed elimination of flow circulation in 1 x 10 mm² dye cell for the same boundary conditions as simulation results shown in fig 4.10.

Fig 4.10: Flow velocity vectors for binary solvent (50: 50 Ethanol & Glycerol) with 2 m/s inlet flow velocity of 10 x 1 x 70 mm dye cell with tubular entry.

The pressure drop across the dye cell with glycerol is higher (2.4 kg/cm²) than the corresponding pressure drop (1.45 kg/cm²) with ethanol as a solvent for the same boundary condition.
4.14 Boundary Layer Visualization

The simulation results for a range of velocities between 0.2 m/s and 2.5 m/s show boundary layer thickness (99% $U_\infty$) was maximum ~370 µm for inlet flow velocity of 0.2 m/s, while it was minimum ~327 µm for inlet flow velocity of 2.5 m/s. In this thin boundary layer the velocity is small, while the pump power and the velocity fluctuations are large, which severely affects the bandwidth and the wavelength stability of the SLM dye lasers. The flow velocities in the active volume were computed for different inlet flow velocities from 0.2 m/s to 2.5 m/s in steps of 0.2 m/s. It was observed that the maximum flow velocity of 6.5 m/s was obtained for an inlet velocity of 2.5 m/s and the maximum flow velocity of 0.5 m/s was obtained with 0.2 m/s inlet flow velocity. The flow velocity profiles for several inlet flow velocity with ethanol ranging from 0.2 m/s to 2.5 m/s for dye cell geometry of 5 mm x 1 mm x 70 mm have been computed and plotted as shown in figure 4.11.

Fig 4.11: Computationally generated velocity profile for 5x1x70 mm dye cell for different inlet flow velocities. (a) 0.2 m/s, (b) 0.5 m/s, (c) 0.75 m/s, (d) 1.0 m/s, (e) 1.2 m/s, (f) 1.4 m/s, (g) 1.6 m/s, (h) 1.8 m/s, (i) 2.0 m/s and (j) 2.5 m/s
The boundary layer thickness was plotted with Reynolds number for the inlet flow velocities ranging from 0.2 m/s to 2.5 m/s as shown in figure 4.12.

Fig 4.12: Boundary layer thickness and velocity with Reynolds number for 5 x 1 x 70 mm dye cell, (a) velocity (b) boundary layer thickness

The computational results of design – XIV are presented here to show the capabilities of the CFD model. The pressure drop across the dye cell at different inlet flow velocities was plotted as shown in figure 4.13

Fig 4.13: Pressure drop across the dye cell (Design – XIV)
It is observed that the pressure drop across the dye cell increases with increasing inlet flow velocity. The static pressure decreases as the flow area decreases from 35 mm$^2$ to 5 mm$^2$ and it regains as the flow area increases in the downstream direction. This dye cell has less permanent pressure loss in comparison to the dye cell with straight rectangular channel of 70 mm length. The computational result of pressure drop and recovery for three different inlet flow velocities is shown in figure 4.13.

### 4.15 Dye Cell Design Considerations

The dye cell used as active volume for dye laser is one of the vital components, which facilitate single mode operation at high repetition rate. The dye cell design has to be such that the flow channel in the dye laser reduces thermal and pressure variations in the flowing dye solution particularly in the vicinity of the active volume. The pump power and repetition rate are related to each other as increasing repetition rate demands increase in the velocity in the flow channel. The higher flow velocity in the dye flow channel results in self-heating due to wall friction. Eddies due to turbulence resulting in disturbances to the streamline and local cavitation in the solvent are some of the limiting conditions for increasing the flow velocity for a high repetition rate laser. The dye cell physical dimensions are very important for single mode operation as larger physical dimensions increase the cavity length, resulting in a smaller cavity FSR, which demands higher resolution from the grating for selecting one longitudinal mode without beam expanders. A judicious design of the dye cell flow channel having smaller physical dimensions and appropriate selection of active volume and pump powers helped solve this problem.

### 4.16 Fabrication of Dye Cells

Two dye cells were fabricated and the simulation results were verified by operating them in the SLM dye laser with high repetition rate (~ 9 kHz) CVL as the pump source. The dye cell is made of two stainless steel (SS) pieces precisely cut (wire EDM cut from a single SS block), polished (diamond polished surfaces) and welded together to form the flow channel of dimensions 10 mm x 1 mm x 70 mm and 5 mm x 1 mm x 70 mm. The dye solution flow direction was from top to bottom for both the dye cells. The C channels were engraved on two SS plates by wire cutting and they were welded together to form the flow channel. The top and bottom pieces were made
from two SS blocks of required dimensions. The 6 mm OD and 4 mm ID SS tube of
30 mm length were welded at each SS block. The orientation can be seen from the
fittings on the length side of SS block a rectangular slot was created by spark erosion
process up to 5mm depth to accommodate the dye cell flow channel. A through hole
of 4 mm ID was made along the length of the slot to match the flow channel area of
10 mm$^2$ of the dye cell with flow dimensions of 10 x 1x 70 mm. While for 5 x 1x 70
mm dye cell a rectangular slot of 5 mm$^2$ was created by spark erosion method to
match the flow cross-sectional area of 5 mm$^2$. The 4 mm ID flexible polyurethane
tubing was used for circulating dye solution through the dye cells.

A photograph of the fabricated dye cell with flow channel of 10 mm x 1 mm x 70 mm
is shown in figure 4.14. Two optical quality broadband AR coated quartz windows
($\sim \lambda/4$) of diameter 20 mm and 2 mm thickness are fixed on the dye cell with Viton
O rings, maintaining flow cross section area of 10 mm$^2$. This dye cell was made with
tubular entry of 4 mm internal diameter tube to match the cross sectional area of flow
channel with the area of flow entry. The flow channel was polished to avoid the flow
disturbances in the stream.

![Flow Channel: 10 X 1 X 70 mm](image)

Fig 4.14: Photograph of the SLM stainless steel dye cell (10 x 1 x 70 mm)
Two optical quality broadband AR coated quartz windows ($\sim \lambda/4$) of 10 mm diameter and 2 mm thickness are fixed on the dye cell of internal flow channel width of 5 mm x 1 mm x 70 mm with Viton O-ring seal. The fluid enters into the flow channel through a slit of 5 mm x 1 mm area for the 5 mm x 1 mm x 70 mm dye cell. The photograph of the dye cell with flow channel of 5 mm x 1 mm x 70 mm is shown in figure 4.15. The performance of these two dye cells was experimentally tested by operating them in the single mode condition with the SLM dye laser system as described below.

4.17 Testing of Dye Cells with SLM Dye Laser

The simulation results with ethanol medium (viscosity 1.078 Cp and density 789 kg/m$^3$) for the dye flow channel of dimensions 10 x 1 x 70 mm was not able to demonstrate stable SLM operation with 9 kHz pulse repetition rate CVL pumping. There was frequent switching of modes from single mode to two modes for this dye laser. When this laser was operated with binary solvent, a stable single mode
operation was established and the bandwidth of this laser was measured to be \( \sim \) 650 MHz. The SLM dye cell 5 mm x 1 mm x 70 mm was characterized with a flow system. The flow velocity in the dye cell was varied using a variable frequency drive (VFD) and flow characteristics were plotted as shown in figure 4.16.

Fig 4.16: The line pressure versus flow velocity for the SLM dye cell.

A stable single mode operation with low viscosity solvent such as ethanol was established on reduction of the cross section area of the dye cell. A minimum time averaged line width of \( \sim \) 375 MHz was obtained with this dye cell. This line width is \( \sim \) 40% less than bandwidth obtained with binary solvent of 650 MHz for 10 mm\(^2\) cross section dye cell. The dye flow control was done with variable frequency drive (VFD) to change the Reynolds number and thereby improve the passive frequency stability for the present SLM dye laser. The detailed characterization of the SLM dye laser is presented in Chapter – V of this thesis.
References


