CHAPTER – 4

THREE DIMENSIONAL CFD ANALYSIS OF CENTRIFUGAL FAN

4.1 Introduction

Computational Fluid Dynamics (CFD) is the use of computers and numerical techniques together to solve problems involving fluid flow. CFD has been successfully applied in many areas of fluid mechanics. This includes aerodynamics of cars and aircraft, hydrodynamics of ships, pumps and turbines and thermodynamics of fan, blower, and compressor. Turbomachines are combination of stationary and rotating passages. Fluid passing through these passages either gets or releases energy. Turbo machinery flows are naturally unsteady mainly due to the relative motion of rotors and stators and the natural flow instabilities present in tip gaps and secondary flows. There are many methods to simulate stationary and rotating passages together by solving their mass, momentum and energy transfer equations. Computational fluid dynamics is one of the most powerful tools to simulate such complex flow system. CFD gives feasible solution of the problem which is under hazardous conditions in real. It also extends its solution beyond the workable boundaries of the problem. It allows checking solution optimally by offering wide range of variable parameters for a problem under study.

Any realistic flow simulation has to be done on a three-dimensional basis. Nowadays virtual three dimensional flow analyses is feasible and allowing designer to have better estimate of influence of spatial parameters on performance of the turbo machine. Computational fluid dynamics (CFD) provides an accurate alternative to scale model testing, with variations available in simulation parameters due to recent advances in computing power, together with powerful graphics and interactive three dimensional manipulations of models. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world.

Centrifugal fan performance can truly be ascertained by experimental evaluations, but CFD analysis can greatly help in reducing number of experimental
iterations. CFD can also help to understand profile distribution of mass flow, pressure and velocity at infinitesimal planes of centrifugal fan geometry under study. This could not be possible merely by experiments and hence CFD analysis and experimental evaluation are equally important and mutually exclusive.

In the present course of work, backward and forward curved radial tipped centrifugal fans are designed as per unified design methodology and simulated using three dimensional computational fluid dynamic (CFD) approach. This numerical analysis is carried out on designed geometry under varying number of blades and speed of impeller rotation at design and off-design conditions. Flow is simulated by considering non-viscous, incompressible by using finite volume approach. Initializing conditions were given to different cases under study by varying their mass flow rate, rotational speed and number of blades. This three dimensional CFD analysis is carried out by using ANSYS’ GAMBIT and FLUENT software by using ‘Reynolds-averaged Navier-Stokes’ equations (RANS) and ‘Realizable k-ε model’. ‘Standard’ wall function is used to resolve wall flows and ‘Simple’ algorithm is used for coupling pressure and velocity [113, 118 and 121].

It is necessary to mention over here that many researchers have categorically opined to go for complete modeling of fan rather than segmental approach to achieve more realistic flow physics across the whole fan [29, 66, 108, 115, 117, 120 and 121]. Accordingly in the present work the complete modeling of both backward curved radial tipped fan (BCRT) and forward curved radial tipped fan (FCRT) designed as per proposed unified design methodology is presented.

Further it is worth to emphasize over here that the basic purpose of this 3-D CFD study is to establish the numerical validity of proposed unified design methodology to offer design point performance.

4.2 Moving Reference Frame (MRF)

To impose rotational field in any turbomachinery component, moving reference frame (MRF), mixing plane interface model and sliding mesh approaches are used [121]. The MRF is a “non-physical snapshot approach” which cannot be compared to a snapshot of a transient simulation, because the solution “does not know anything about what happened before”. The MRF approach utilizes relative motion between moving and stationary zones to transmit calculated values of flow
parameters. The mixing plane model on the other hand has the advantage that only one pitch of the impeller has to be modeled to reduce computational efforts. This approach would not allow investigating different states of the impeller pitch depending on its circumferential location. Sliding mesh is explicitly used for transient simulation. It requires advance computational resources and prolonged time to converge the solution. In present study, the computational domain is asymmetric circumferentially and steady flow and very low pressure ratio conditions are prevailing. Hence MRF approach is used and shadow wall method is used at interior surface to create continuous flow path between moving and stationary zones [121].

FLUENT's moving reference frame modeling capability allows to model problem involving moving parts in selected cell zones. Such problems typically involve moving parts (such as rotating blades, impellers, and similar types of moving surfaces), and flow around these moving parts. When a moving reference frame is activated, the equations of motion are modified to incorporate the additional acceleration which occurs due to transformation from stationary to the moving reference frame. By solving these equations in a steady-state manner, the flow around the moving parts can be modeled. In most cases, the moving parts make the problem unsteady when viewed from the stationary frame.

The principal reason for employing a moving reference frame for impeller in centrifugal fan is to change a problem which is unsteady in the stationary (inertial) frame and steady with respect to the moving frame. For a steady rotating frame (i.e., the rotational speed is constant), it is possible to transform the equations of fluid motion to the rotating frame such that steady-state solutions are possible. By default, FLUENT permits the activation of a moving reference frame with a steady rotational speed. If the rotational speed is not constant, the transformed equations will contain additional terms which are not included in FLUENT's formulation (although they can be added as source terms by using user-defined functions).

4.2.1 Equations for a moving reference frame

A coordinate system which is rotating steadily with angular velocity \( \dot{\vartheta} \) relative to a stationary (inertial) reference frame is illustrated by Figure 4.1. The origin of the rotating system is located by a position vector \( \vec{r}_0 \). The axis of rotation is defined by a unit direction vector \( \vec{\hat{a}} \) such that

\[
\vec{\dot{\vartheta}} = \omega \vec{\hat{a}}
\]
The computational domain for the CFD problem is defined with respect to the rotating frame such that an arbitrary point in the CFD domain is located by a position vector \( \vec{r} \) from the origin of the rotating frame. The fluid velocities can be transformed from the stationary frame to the rotating frame by using the following relation:

\[
\vec{v}_r = \vec{v} - \vec{u}_r
\]

Where

\[
\vec{u}_r = \omega \times \vec{r}
\]

In the above equations, \( \vec{v}_r \) is the relative velocity (the velocity viewed from the rotating frame), \( \vec{v} \) is the absolute velocity (the velocity viewed from the stationary frame), and \( \vec{u}_r \) is the “whirl” velocity (the velocity due to the moving frame).

When the equations of motion are solved in the rotating reference frame, the acceleration of the fluid is augmented by additional terms that appear in the momentum equations.

![Stationary and Rotating Reference Coordinate System](image)

**Figure 4.1 Stationary and Rotating Reference Coordinate System [118]**
4.2.2 Absolute velocity formulation

[66]

For the absolute velocity formulation, the ‘Reynolds-averaged Navier-Stokes’ equations (RANS) are the governing equations for a steady rotating fluid flow. For conservation of mass it is:

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \]

For conservation of momentum it is:

\[ \frac{\partial}{\partial t} \left( \rho \vec{v} \right) + \nabla \cdot (\rho \vec{v} \times \vec{v}) + \rho \vec{P} = -\nabla p + \nabla \vec{T} + \vec{F} \]

For conservation of energy it is:

\[ \frac{\partial}{\partial t} \left( \rho E \right) + \nabla \cdot (\rho \vec{v} E + p \vec{v}) = \nabla \cdot (k \nabla T + \vec{v} \cdot \vec{v}) + S_h \]

The multiple moving reference frame (MMRF) is a steady-state approximation in which individual cell zones move at different rotational and/or translational speeds. The flow in each moving cell zone is solved using the moving reference frame equations. If the zone is stationary (\( \omega = 0 \)), the stationary equations are used. At the interfaces between cell zones, a local reference frame transformation is performed to enable flow variables in one zone to be used to calculate fluxes at the boundary of the adjacent zone.

It should be noted that the MRF approach does not account for the relative motion of a moving zone with respect to adjacent zones (which may be moving or stationary); the grid remains fixed for the computation. This is analogous to freezing the motion of the moving part in a specific position and observing the instantaneous flow field with the rotor in that position. Hence, the MRF is often referred to as the “frozen rotor approach.” [127].

While the MRF approach is clearly an approximation, it can provide a reasonable model of the flow for many applications. For example, the MRF model can be used for turbomachinery applications in which rotor-stator interaction is relatively weak, and the flow is relatively uncomplicated at the interface between the moving and stationary zones. It can be used for mixing tank problems where impeller-baffle interactions are also relatively weak.
4.2.3 Limitations of MRF approach

The interfaces separating a moving region from adjacent regions must be oriented such that the component of the frame velocity normal to the boundary is zero. That is, the interfaces must be surfaces of revolution about the axis of rotation defined for the fluid zone. For a translational moving frame, the moving zone's boundaries must be parallel to the translational velocity vector.

Multiple moving reference frames is meaningful only for steady flow. However, FLUENT will allow to solve an unsteady flow when multiple reference frames are being used. In this case, unsteady terms are added to all the governing transport equations. For unsteady flows, a sliding mesh approach will yield more meaningful results than an MRF calculation.

Particle trajectories and path lines have drawn by FLUENT uses the velocity relative to the cell zone motion. For mass-less particles, the resulting path-lines follow the streamlines based on relative velocity. Translational and rotational velocities are assumed to be constant (time varying $\omega$, $v_t$ are not allowed).

4.3 Description of Solution Algorithm

Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations. These are computationally less expensive to solve it. However, the modified equations contain additional unknown variables, and additional turbulence models are needed to determine these variables in terms of known quantities.

FLUENT provides the following choices for turbulence models:
- Spalart-Allmaras model,
- $k-\varepsilon$ models,
- Standard $k-\varepsilon$ model,
- Renormalization-group (RNG) $k-\varepsilon$ model,
Realizable $k$-$\varepsilon$ model, 
3 phase $k$-$\omega$ models and 
Shear-stress transport (SST) $k$-$\omega$ model.

Any single turbulence model stated above is not universally accepted as superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. The simplest "complete models" of turbulence are two equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard $k$-$\varepsilon$ model falls within this class and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [139]. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empirism. As the strengths and weaknesses of the standard $k$-$\varepsilon$ model have become known, improvements have been made to the model to improve its performance. RNG $k$-$\varepsilon$ model and the realizable $k$-$\varepsilon$ model are two of these variants available in FLUENT. The standard $k$-$\varepsilon$ model is a semi empirical model based on model transport equations for the turbulence kinetic energy ($k$) and its dissipation rate ($\varepsilon$). The model transport equation for $k$ is derived from the exact equation, while the model transport equation for $\varepsilon$ was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart.

The realizable $k$-$\varepsilon$ is a relatively recent developed model and differs from the standard $k$-$\varepsilon$ model in two important way, it contains a new formulation for the turbulent viscosity and a new transport equation for the dissipation rate $\varepsilon$, has been derived from an exact equation for the transport of the mean square vorticity fluctuation. It is due to this fact that in the present work, realizable $k$-$\varepsilon$ model is used.
4.3.1 Realizable k-ε model

[118]

The term “realizable” means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard k-ε model nor the RNG k-ε model is realizable. An immediate benefit of the realizable k-ε model is that it more accurately predicts the spreading rate of both planar and round jets. It is also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation.

Both the realizable and RNG k-ε models have shown substantial improvements over the standard k-ε model where the flow features include strong streamline curvature, vortices and rotation. Since the model is still relatively new, it is not clear in exactly which instances the realizable k-ε model consistently outperforms the RNG model. However, initial studies have shown that the realizable model provides the best performance of all the k-ε model versions for several validations of separated flows and flows with complex secondary flow features [66, 121].

One limitation of the realizable k-ε model is that it produces non-physical turbulent viscosities in situations when the computational domain contains both rotating and stationary fluid zones (e.g., multiple reference frames, rotating sliding meshes). This is due to the fact that the realizable k-ε model includes the effects of mean rotation in the definition of the turbulent viscosity. This extra rotation effect has been tested on single rotating reference frame systems and showed superior behavior over the standard k-ε model [66].

This model has been extensively validated for a wide range of flows, including rotating homogeneous shear flows, free flows including jets and mixing layers, channel and boundary layer flows, and separated flows. For all these cases, the performance of the model has been found to be substantially better than that of the standard k-ε model. The realizable k-ε model resolves the round-jet anomalies by predicting the spreading rate for axi-symmetric jets as well as planar jets [66, 121].
4.4 Computational Grid

ANSYS inc. offers Gambit as pre processor software. The pre-processing work in present course is done by using GAMBIT (Geometry and Mesh Building Intelligent Toolkit) software. GAMBIT is very robust tool for geometry generation and it gives total user control while doing meshing work. Allocation of nodes as per design geometry is carried out. The co-ordinates of each node are spatially created and then two dimensional lines and curves are generated. Later faces from lines and then volumes from faces are generated. Volumes are identified according to required moving and stationary zones. To create number of blades, first one blade profile is generated and then it is copied around the z-axis in required numbers as per design. After generating blade volumes, they are subtracted from impeller zone. For this reason the radial and axial size of impeller is kept little higher than blades’ volume. Adjacent stationary and moving boundaries are differentiated by using shadow wall interface concept. To create ‘shadow wall’ interface between impeller and casing, the volume of impeller is ‘spitted’ from volume of scroll casing. Each moving and stationary zone is given property of fluid.

Due to complex flow structure in centrifugal fan the unstructured/hybrid grid with tetrahedral element is recommended by few researchers [117, 118, 124]. Accordingly in present work unstructured grid/hybrid grid with tetrahedral elements is selected. Here finer elements in impeller region are used as compared to inlet nozzle and scroll casing as generated by commercial code GAMBIT [117].

4.4.1 Grid independency test

Grid independency is the process to find out optimum number of grids/elements. Simulation is carried out on complete model of fan with volute and similar geometry and boundary conditions with varying number of mesh elements.

To optimize number of grids/elements and to develop grid independency, first very coarse grids are generated and then successively decreasing the size of mesh elements, results are evaluated. The parameters under study are: change in static pressure across the fan, static outlet pressure and overall pressure ratio of the fan. Results obtained for varied number of total elements are presented in Table 4.1. Its graphical representation is given in Figure 4.2.
Table 4.1 Grid Independency Test

<table>
<thead>
<tr>
<th>Mesh</th>
<th>No. of Mesh Elements</th>
<th>∆P Static (Pa)</th>
<th>Static Outlet Pressure (Pa)</th>
<th>Pressure Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>219210</td>
<td>797.665</td>
<td>657.97</td>
<td>1.00788</td>
</tr>
<tr>
<td>b</td>
<td>419633</td>
<td>834.69</td>
<td>681.59</td>
<td>1.00825</td>
</tr>
<tr>
<td>c</td>
<td>869068</td>
<td>834.24</td>
<td>679.67</td>
<td>1.00824</td>
</tr>
<tr>
<td>d</td>
<td>1934222</td>
<td>833.37</td>
<td>679.04</td>
<td>1.008237</td>
</tr>
</tbody>
</table>

Figure 4.2 Graphical Representation of Mesh Independency Test

In present case, it is clearly seen from Figure 4.2 and Table 4.1 that mesh ‘b’ results are more accurate as compared to mesh ‘a’ result. Further it is found that the relative deviation of the static pressure rise, static outlet pressure and pressure ratio between mesh ‘b’ and ‘c’ is less than 1%. Mesh ‘c’ and ‘d’ yield similar results.
Further increase in mesh elements will not affect accuracy, simulation results. Hence mesh ‘b’ with 419633 mesh elements is accepted for all further simulations to save simulation time.

Figure 4.3 shows the wire-frame while Figure 4.4 shows shaded diagram and Figure 4.5 and Figure 4.6 shows two and three dimensional mesh diagram of centrifugal fan under study with 419633 mesh elements for volume discretization.
4.5 Simulation Parameters Used by Solver

ANSYS FLUENT software is used for post processing work. It contains broad physical modeling capabilities to model flow, turbulence and heat transfer phenomenon. This is very essential aspect for turbomachinery simulation. It provides multiple choices in solver option for a wide range of speed regimes.

Efficient energy transfer in a centrifugal fan depends upon good impeller inlet conditions, proper blade profile, gradual change in scroll casing area and overall smooth surface finish. For such energy transfer, flow lines must be parallel to each other and should generate streamlined flow within three dimensional guided passages [7, 92].
The inlet nozzle and scroll casing are stationary zones and impeller is a moving zone. Being steady flow and very low pressure ratio in this case, moving reference frame (MRF) approach is used to impose rotational field to impeller zone of centrifugal fan[113]. The blades are moving with a same rotational velocity as impeller zone by giving moving wall condition with zero relative rotational speed to adjacent cell zone. Shadow wall method is used as interior surface to create continuous flow path between moving and stationary zones.

In the centrifugal fans, the three-dimensional motion of the air is thought to be the incompressible and steady flow. Three dimensional simulation is carried out using ‘Reynolds-averaged Navier-Stokes’ equations (RANS) and ‘Realizable \( k-\varepsilon \) model’. As the fluid in a state of turbulence, the Realizable \( k-\varepsilon \) model was selected as the turbulence model to give superior performance for flows involving rotation, separation and recirculation. The ‘standard’ wall function is used to resolve wall flows. The ‘SIMPLE’ algorithm is used for coupling pressure and velocity [118, 122, 112]. Turbulent kinetic energy and dissipation of turbulence uses function of second-order discrete upwind.

Mass flow rate at nozzle inlet is used as inlet boundary condition. Zero gradient outflow condition is used at casing outlet. This is done for fully developed flow conditions. At inlet boundary condition, 5% turbulent intensity and 0.5 turbulent length scale is applied [118]. This is calculated based on cube root of domain volume and used for turbulence specifications. ‘No-slip’ boundary condition is used for all walls. The discharge at nozzle inlet at each step of rotational speed of impeller is varied by varying input boundary condition of inlet mass flow rate.

Present simulation work is carried out on the backward and forward curved radial tipped centrifugal fans designed as per unified design methodology, as explained in chapter 3. The factors that impact fan’s performance are coupled with each other. Various factors have a combined action together on the fan performance. This work takes the efficiency \( \eta \) as a maximizations goal, and takes the number of blade \( Z \), speed of impeller rotation \( N \) and the volume flow rate \( Q \) as the variable quantity, and constructs the optimized mathematical model:

\[
Max \ f(x, y, z)
\]

In the model, \( f(x, y, z) \) represents the objective function of efficiency \( \eta \), and parameters \( x, y \) and \( z \) separately represent the number of blade, speed of rotation and
the volume flow for backward and forward curved radial tipped centrifugal fans designed as per unified design methodology.

Figure 4.7 shows consolidated algorithms for overall work carried out during the course of this work.

![Figure 4.7 Consolidated Algorithms for Simulated Cases](image)

The discretization of pressure is done by using ‘PRESTO’ scheme. This scheme is generally used to obtain good results if there is vortex flow field. The momentum, turbulent kinetic energy, turbulent dissipation rate and energy are discretized by using ‘second order upwind method’. Maximum residuals are less than $10^{-5}$ [117, 118, 122] as solution convergence criterion.

Initializing conditions were given to different cases under study by varying their mass flow rate, rotational speed and number of blades for BCRT and FCRT fans. The results of this steady simulation (MRF approach) give better understanding of flow behavior and flow visualization inside the centrifugal fans which are discussed in subsequent sections.
4.6  Numerical Flow Analysis of Backward Curved and Forward Curved Radial Tipped Blade Centrifugal Fan

Qualitative and quantitative simulation result analysis of the flow in radial tipped backward curved blade (BCRT) and forward curved blade (FCRT) centrifugal fan with varying discharge, speed and number of blades is presented herein. Discharge is varied between 0.1 to 0.6 m³/s with the increment of 0.1 m³/s. Speed of rotation is varied as 2500, 2650 and 2800 rpm, while number of blades is varied as 12, 16 and 24 for each case stated above.

Efficient energy transfer in a centrifugal fan depends upon proper blade profile, gradual change in area of scroll casing and smooth surface finish within impeller vane passage. Flow lines must be parallel to each other and should generate streamlined flow within three dimensional passages [13, 26].

Figure 4.8 and 4.9 shows flow streamlines at mid plane cross-section of a centrifugal fan under study with 16 numbers of blades for BCRT and FCRT fans, respectively at 2800 rpm. Here streamlines are seen parallel and efficiently guided within entire flow passage. It confirms that flow across the stage is well guided and flow leaves impeller smoothly and enters in scroll casing without circulation. Scroll casing progressively guides flow to outlet section of the centrifugal fan.

Little flow re-circulation is observed near tongue region and inter blade passage. Small vortices are observed within blade passages and when it leaves tongue region. The vortices start to form near the entrance of the blade passages. Then they develop along the passage and gradually move to the blade suction side. Their effect becomes weaker near the outlet of the blade passage. These observations are quite in tune with the few results available in published literature [46, 61, 115] and thus validates the present approach of 3-D CFD simulation.
In general, the flow in all passages seems to be reasonably smooth with 16 numbers of blades. Similar kind of streamlined flow is observed within fan assembly for all other cases under simulation having variable parameters of number of blades, speed of rotation and mass flow rate.

### 4.6.1 Flow visualization in centrifugal fan passage

Figures 4.10 to 4.12 shows static pressure, total pressure and velocity magnitude contours on mid plane profile along Z-axis for flow visualization in backward curved radial tipped blade (BCRT) centrifugal fan. While, Figures 4.13 to 4.15 presents similar fan contours, for forward curved radial tipped blade (FCRT) centrifugal fan, respectively.

Static pressure, total pressure and velocity contour plots are obtained in each case of rotational speed in the step of N = 2500, 2650 and 2800 rpm and set of number of blades Z = 12, 16 and 24. Here discharge is kept constant at design point value of $Q = 0.5 \text{ m}^3/\text{s}$. Other fan parameters and boundary conditions are also kept constant. Every case legend is plotted with maximum and minimum values. This can help to get visual comparison at a glance.
Chapter – 4: Three Dimensional CFD Analysis of Centrifugal Fan

BCRT Fan at 2500 rpm and Q=0.5 m$^3$/s

Range for static pressure contour maxima at 676 Pa
Range for total pressure contour maxima at 1533 Pa
Range for velocity magnitude contour maxima at 60 m/s

<table>
<thead>
<tr>
<th></th>
<th>Z=12</th>
<th>Z=16</th>
<th>Z=24</th>
</tr>
</thead>
<tbody>
<tr>
<td>Static Pressure Contours</td>
<td><img src="image1" alt="Static Pressure Contours Z=12" /></td>
<td><img src="image2" alt="Static Pressure Contours Z=16" /></td>
<td><img src="image3" alt="Static Pressure Contours Z=24" /></td>
</tr>
<tr>
<td>Total Pressure Contours</td>
<td><img src="image4" alt="Total Pressure Contours Z=12" /></td>
<td><img src="image5" alt="Total Pressure Contours Z=16" /></td>
<td><img src="image6" alt="Total Pressure Contours Z=24" /></td>
</tr>
<tr>
<td>Velocity Contours</td>
<td><img src="image7" alt="Velocity Contours Z=12" /></td>
<td><img src="image8" alt="Velocity Contours Z=16" /></td>
<td><img src="image9" alt="Velocity Contours Z=24" /></td>
</tr>
</tbody>
</table>

Figure 4.10 Static Pressure, Total Pressure and Velocity Contours for BCRT Fan at Q= 0.5 m$^3$/s, N=2500 rpm and Z=12, 16, 24
BCRT Fan at 2650 rpm and Q=0.5 m$^3$/s
Range for static pressure contour maxima at 697 Pa
Range for total pressure contour maxima at 1817 Pa
Range for velocity magnitude contour maxima at 58 m/s

Figure 4.11 Static Pressure, Total Pressure and Velocity Contours for BCRT Fan at Q= 0.5 m$^3$/s, N=2650 rpm and Z=12, 16, 24
BCRT Fan at 2800 rpm and $Q=0.5 \, \text{m}^3/\text{s}$

Range for static pressure contour maxima at 920 Pa

Range for total pressure contour maxima at 2010 Pa

Range for velocity magnitude contour maxima at 58 m/s

Figure 4.12 Static Pressure, Total Pressure and Velocity Contours for BCRT Fan at $Q=0.5 \, \text{m}^3/\text{s}$, $N=2800$ rpm and $Z=12, 16, 24$
### FCRT Fan at 2500 rpm and $Q=0.5 \text{ m}^3/\text{s}$

- Range for static pressure contour maxima at 840 Pa
- Range for total pressure contour maxima at 1690 Pa
- Range for velocity magnitude contour maxima at 50 m/s

<table>
<thead>
<tr>
<th>Z=12</th>
<th>Z=16</th>
<th>Z=24</th>
</tr>
</thead>
</table>

#### Static Pressure Contours

![Static Pressure Contours](image)

#### Total Pressure Contours

![Total Pressure Contours](image)

#### Velocity Contours

![Velocity Contours](image)

**Figure 4.13** Static Pressure, Total Pressure and Velocity Contours for FCRT Fan at $Q=0.5 \text{ m}^3/\text{s}$, $N=2500$ rpm and $Z=12, 16, 24$
FCRT Fan at 2650 rpm and $Q=0.5 \text{ m}^3/\text{s}$

Range for static pressure contour maxima at 1010 Pa
Range for total pressure contour maxima at 1880 Pa
Range for velocity magnitude contour maxima at 53 m/s

Figure 4.14 Static Pressure, Total Pressure and Velocity Contours for FCRT Fan at $Q=0.5 \text{ m}^3/\text{s}$, $N=2650 \text{ rpm}$ and $Z=12, 16, 24$
Chapter – 4: Three Dimensional CFD Analysis of Centrifugal Fan

FCRT Fan at 2800 rpm and $Q=0.5 \text{ m}^3/\text{s}$
Range for static pressure contour maxima at 1190 Pa
Range for total pressure contour maxima at 2090 Pa
Range for velocity magnitude contour maxima at 55 m/s

Here it is seen from Figure 4.10 to 4.15 that at low rotational speed of impeller, the static and total pressure generated by centrifugal fan is low. It increases with respect to speed of rotation. Static pressure, total pressure and velocity vector across the fan stage attain maxima of 920 Pa, 2010 Pa and 58 m/s, respectively for BCRT fan, while static pressure, total pressure and velocity vector across the fan stage attain maxima of 1190 Pa, 2090 Pa and 55 m/s, respectively for FCRT fan at design speed of 2800 rpm. High pressure regions and increase in velocities is seen...
around impeller outlet. Pressure and velocity increases with increase in impeller speed and number of blades. Growth in velocity magnitude in impeller blade passages is clearly seen. Simulated results reveals that 834.7 Pa average static pressure rise is observed for the BCRT and 1106.7 Pa for the FCRT centrifugal fan for 16 number of blades across the stage. This shows 15% and 11% deviation between design point static pressure rise of 981.2 Pa for BCRT and FCRT fan, respectively, by keeping all other parameters constant.

4.6.2 Flow visualization on blade surfaces and velocity vectors

Post-processing three dimensional simulation results are presented in Figure 4.16 to 4.18 for flow visualization on suction and discharge side blade surfaces of BCRT fan and velocity vectors within entire flow passage, while, Figures 4.19 to 4.21 presents similar three dimensional simulation for forward curved radial tipped blade (FCRT) centrifugal fan, respectively. It is represented in the form of contours of static pressure, velocity magnitude, turbulent kinetic energy and velocity vectors. Various contour plots are obtained for rotational speed of 2800 rpm and number of blades in the steps of 12, 16 and 24. Here discharge is kept constant at design value of Q=0.5 m$^3$/s.

![Static Pressure Contours](image1)

![Velocity Magnitude](image2)

![Turbulent Kinetic Energy](image3)

![Velocity Vectors](image4)

**Figure 4.16 Static Pressure, Velocity Magnitude, Turbulent Kinetic Energy and Velocity Vector Contours for BCRT Fan at Q= 0.5 m$^3$/s, N=2800 rpm and Z=12**
Chapter – 4: Three Dimensional CFD Analysis of Centrifugal Fan

Figure 4.17 Static Pressure, Velocity Magnitude, Turbulent Kinetic Energy and Velocity Vector Contours for BCRT Fan at Q = 0.5 m$^3$/s, N=2800 rpm and Z=16

Figure 4.18 Static Pressure, Velocity Magnitude, Turbulent Kinetic Energy and Velocity Vector Contours for BCRT Fan at Q = 0.5 m$^3$/s, N=2800 rpm and Z=24
Figure 4.19 Static Pressure, Velocity Magnitude, Turbulent Kinetic Energy and Velocity Vector Contours for FCRT Fan at Q = 0.5 m$^3$/s, N=2800 rpm and Z=12

Figure 4.20 Static Pressure, Velocity Magnitude, Turbulent Kinetic Energy and Velocity Vector Contours for FCRT Fan at Q = 0.5 m$^3$/s, N=2800 rpm and Z=16
Based on the flow visualization study from Figure 4.16 to 4.21, it is seen that static pressure contours on suction and discharge side of blade surfaces indicates existence of wide pressure difference. Static pressure is very much reduced on interior surface of back shroud plate near hub region. This is due to turning of flow from axial to radial direction and lack of flow guidance available in that wall region [45, 138]. Increase in velocity magnitudes can be seen at impeller tip accompanied by static pressure rise due to increase in fluid energy level imparted by rotating impeller [26]. Uniform turbulent kinetic energy dissipation confirms the reliability of realizable k-ε model adopted in solution algorithm [66]. Study of velocity vectors clearly shows streamlined flow in entire flow region. Some flow circulation is observed near tongue region which indicates a need of better scroll tongue design. Small vortices are also seen within blade passages. As number of blades increases, flow gets efficiently guided and vorticity within blade passage reduces and magnitude of velocity increases [61].
4.6.3 Three dimensional flow visualization

Figure 4.22 to 4.24 shows three dimensional flow field visualization on various Y-Z planes along X-axis for BCRT centrifugal fan, while Figure 4.25 to 4.27 shows three dimensional flow field visualization for FCRT centrifugal fan. These flow fields represent static pressure, total pressure and velocity magnitude contours at $Z=16$, $N=2800$ and $Q=0.5 \text{ m}^3/\text{s}$. The length of velocity vectors represents velocity magnitude and direction of flow at that particular point.

Figure 4.22 Static pressure contour planes along the x-axis for Backward curved blades with $Z=16$, $N=2800$ and $Q=0.5 \text{ m}^3/\text{s}$

Figure 4.23 Total pressure contour planes along the x-axis for Backward curved blades with $Z=16$, $N=2800$ and $Q=0.5 \text{ m}^3/\text{s}$
Figure 4.24 Velocity magnitude contour planes along the x-axis for Backward curved blades with Z=16, N=2800 and Q=0.5 m³/s

Figure 4.25 Static pressure contour planes along the x-axis for Forward curved blades with Z=16, N=2800 and Q=0.5 m³/s

Figure 4.26 Total pressure contour planes along the x-axis for Forward curved blades with Z=16, N=2800 and Q=0.5 m³/s
Figure 4.22 and Figure 4.25 shows static pressure contours at different Y-Z planes along X axis for BCRT and FCRT fans. At impeller eye, impeller inlet, impeller outlet and scroll casing outlet, it measures -153 Pa, -145 Pa, 41 Pa and 681 Pa for BCRT fans and -170 Pa, -148 Pa, 354 Pa and 937 Pa, respectively for FCRT fans. Negative static pressure is observed at impeller eye and inlet as velocity increases in this area due to suction created by impeller. Static pressure rise within impeller passage is observed. Static pressure also rises progressively between impeller exit to scroll casing exit. It confirms efficient diffusion of flow within scroll casing [140].

Figure 4.23 and Figure 4.26 shows uniform distribution of total pressure near all inside boundaries of scroll casing for BCRT and FCRT fans, respectively. This is happening due to velocity diffusion and rises in static pressure which is distributed evenly at scroll casing periphery. Total pressure is divided in to two zones at scroll casing exit near tongue region. This is due to obstruction of recirculation shield provided as per unified design methodology. Recirculation shield reduces scroll casing area and increases magnitude of leaving velocity and hence minor reduction in total pressure is observed [137].

Figure 4.24 and Figure 4.27 shows velocity magnitude contour planes along the X-axis for backward curved blades and forward curved blades fan, respectively with $Z = 16$, $N = 2800$ and $Q = 0.5 \text{ m}^3/\text{s}$. Magnitude of high velocity is seen near
impeller exit. This phenomenon occurs as fluid leaving the impeller gets energy and increases its velocity and static pressure level. Thereafter velocities reduce within scroll casing. Again two zones of velocities of different magnitudes are observed near scroll casing exit due to obstruction of recirculation shield [137].

4.7 Overall Performance Prediction

Centrifugal fans with forward and backward curved radial blade impellers are simulated for variable parameters using 3-D CFD analysis. Based on this simulation the overall dimensional performance in terms of static pressure rise with respect to volume flow rate is evaluated for 12, 16 and 24 number of blades with flow rate varying in the range of 0.1 m$^3$/s to 0.6 m$^3$/s for both BCRT and FCRT fans and the same is depicted in Figure 4.28 at a typical design speed of 2800 rpm with design point marked at 981.2 Pa static pressure rise and 0.5 m$^3$/s volume flow rate.

Similarly to account for complete variation of speed flow and number of blades, non-dimensional performance characteristics is also obtained in terms of variation of pressure coefficient on as a function of flow coefficient for both BCRT and FCRT fans and the same is presented in Figure 4.29 with design point indication. The observation table for both these simulation results is given in Annexure B.

![Figure 4.28 Static Pressure Rise Vs. Volume Flow Rate for N = 2800 rpm](image-url)
Study of the dimensional and non-dimensional performance characteristics given in Figure 4.28 and 4.29 reveals near ideal characteristics as expected from radial blade impellers with moderate dropping trend [9, 26, 28].

Performance curve are seen outset with increase in number of blades. When number of blades increases up to certain limit, the flow within blades are guided properly, inter blade vorticity reduces and fluid gets more kinetic energy as compared to less numbers of blades. This extra kinetic energy increases the attachment of flow to the blade surface and reduces flow separation effect. Design point lies centrally between forward and backward tipped blades. Performance shows that forward curved blades are useful to run at higher pressure. Similarly from Figure 4.29, it can be said that forward curved blade offers higher pressure coefficient. Large blade angles generates higher flow rate and higher stage pressure rise. Exit whirl component $V_{\alpha 2}$ is large and is leading to a higher stage pressure rise. Such blades have a larger hub to tip ratio which allows large area for the flow entering the stage. Forward curved fans as compared to backward curved develops the higher pressure for a given impeller diameter and speed [28].

Thus, the overall performance as predicated by 3-D CFD simulation clearly establishes the validity of proposed unified design methodology for radial tipped centrifugal fan.
4.8 Closure

The results of numerical simulation for centrifugal fan assuming steady and incompressible flow using MRF approach gives successful insight for flow visualization. After critical evaluation of results obtained from all simulated cases and comparing it with designed and experimental data available for backward and forward curved radial tipped centrifugal fans under variable speed, volume flow rate and numbers of blades, following conclusions are derived.

(i) Energy transfer from impeller to fluid is seen by pressure and velocity contours within all blade passages. Low and high pressure regions along suction and pressure side of a blade are visualized by numerical analysis. Energy transfer from impeller to fluid is also confirmed by pressure and velocity contours within blade passage [14].

(ii) Jet and wakes are observed in the vicinity of tongue region. The flow phenomenon of recirculation near tongue region is confirmed by numerical analysis as shown in stream line diagram given in Figure numbers 4.8 and 4.9. Pressure pulsations are observed at impeller outlet near tongue region. Hence design of tongue is very important to reduce back flow and recirculation. It shows that design of tongue is very much important in fan design to reduce back flow and recirculation. These observations are quite in tune with the observations of [46, 61, 115] and others, and thereby establishes the validity of present 3-D CFD approach.

(iii) The relative velocity in the blade passage becomes more uniform due to proper guidance as number of blades increases, and hence wakes regions decreases. This could reduce noise generated due to wake formation. Formation of wake region is one of the major contributors to the fan losses. Further increase in number of blades would deteriorate the fan performance and boundary layer effects may become dominant [66].

(iv) Static pressure contours at different Y-Z planes along X axis for BCRT fan reveals that, at impeller eye, impeller inlet, impeller outlet and scroll casing outlet, static pressure measures are -153 Pa, -145 Pa, 41 Pa and 681 Pa, respectively. There is 834.7 Pa average static pressure rise is observed for 16 number of blades across the stage. This shows 15% deviation between design
point static pressure rise of 981.2 Pa for 16 numbers of blades and numerical results as obtained by keeping all other parameters constant. While in scroll casing, static pressure rises from 250 Pa to 804 Pa. Rise in static pressure indicates efficient diffusion of flow. Fluid leaves casing at 11.32 m/s velocity.

(v) Static pressure contours at different Y-Z planes along X axis for FCRT fan reveals that, at impeller eye, impeller inlet, impeller outlet and scroll casing outlet, static pressure measures are -170 Pa, -148 Pa, 354 Pa and 937 Pa, respectively. There is 1106.7 Pa average static pressure rise is observed for 16 number of blades across the stage. This shows 11% deviation between design point static pressure rise of 981.2 Pa for 16 numbers of blades and numerical results as obtained by keeping all other parameters constant. While in scroll casing, static pressure rises from 423 Pa to 1131 Pa. Rise in static pressure indicates efficient diffusion of flow. Fluid leaves casing at 11.23 m/s velocity.

(vi) The overall dimensional and non-dimensional performance characteristics as predicted by 3-D CFD analysis for FCRT and BCRT fans with varying flow, speed and number of blades indicate near ideal performance characteristics as expected from radial tipped centrifugal fans. It also shows achievement of design point performance within acceptable limits. Further forward curved radial tipped fan indicates higher static pressure gradient as compared to backward curved centrifugal fan. The nature of curves obtained after simulation study closely follows trend of standard performance curves [9, 26 and 28].

Thus, it may be stated that looking to the uncertainties in 3-D CFD analysis as discussed by [35] and others [113, 121], the respective deviation of 15% and 11% in stage pressure rise at design flow rate of 0.5 m³/s and rotational speed of 2800 rpm for BCRT and FCRT fans with 16 number of blade may be considered as acceptable. Further present 3-D CFD simulation of BCRT and FCRT fans designed as per proposed unified design methodology shows the capabilities of these fans to offer near ideal and design point performance with 16 numbers of blades.

This means that the proposed unified design methodology for radial tipped centrifugal fan may be treated as numerically validated design.