

# A Critical Review on the Application of Computational Fluid Dynamics in Centrifugal Turbomachines

Manjunath L Nilugal<sup>1</sup>, K Vasudeva Karanth<sup>1</sup>, Madhwesh N<sup>1\*</sup>

<sup>1</sup>Department of Mechanical and Manufacturing Engineering, Manipal Institute of Technology, Manipal Academy of Higher Education, Manipal 576104, Karnataka, India

\*Corresponding author: madhwesh.n@manipal.edu

**Abstract:** Centrifugal turbomachines are the popular type of power absorbing machines especially used in many industrial applications. In these devices the energy from the rotating impeller is transferred to the fluid to increase its pressure energy. The flow phenomena in these machines is very complex because of the presence of adverse pressure gradients in the flow passages. The level of complexity of turbomachinery flow fields is directly influenced by flow-path geometry and component blade geometry. Hence there is a dire need of advanced flow modelling techniques to understand and capture the fluid characteristics in these flow devices. To design such flow domains, advanced Computational Fluid Dynamics (CFD) tools are required that are capable of accurately modelling the complex flows encountered in turbomachinery applications. These models reflect the full three-dimensional, turbulent, transonic, and often unsteady nature of the actual flows. In addition, they must allow analyses to be performed in reasonable amounts of time that can be accommodated within a typical design cycle. With the development of fast and validated numerical methods, and the continuous improvement in clock speed of computer processors, highly complex problems are now being solved using CFD methods even more economically and quickly. This article focusses the efficacy of CFD tools in turbomachinery applications and various parametric methodologies pertaining to efficiency improvement of turbomachines in detail.

## 1. Introduction

Turbomachines are the energy exchanging devices. The energy transformation takes place from the rotating blades to flowing fluid with an imposed dynamic action or vice versa. They are classified as power absorbing and power generating machines. The pressure energy of the flowing fluid is increased in the power absorbing machines (centrifugal fans, centrifugal compressors etc.). Power generating machines expands the fluid by reducing its pressure head (Wind turbines, Hydraulic turbines etc.). These machines are further classified based on fluid flow with respect to rotational axis as Axial machines (fluid flows parallel to the axis of impeller rotation) and Radial machines (fluid flows perpendicular to impeller rotation).

Centrifugal turbomachines are classified as power absorbing and radial machines. The fluid enters the inlet and interacts with the rotating impeller to actuate the pressure energy, then the fluid further enters diffuser radially where the kinetic energy is diffused to develop the head. The volute is generally mounted around the diffuser where the major conversion of the energy takes place.

The flow in these devices is very complex and three dimensional because the velocity of the fluid changes at each location and there is presence of the pressure gradients. These pressure fluctuations in interfaces lead to jet-wakes and recirculation zones. The losses occurring in the machines will drastically reduce the efficiency of the machine.

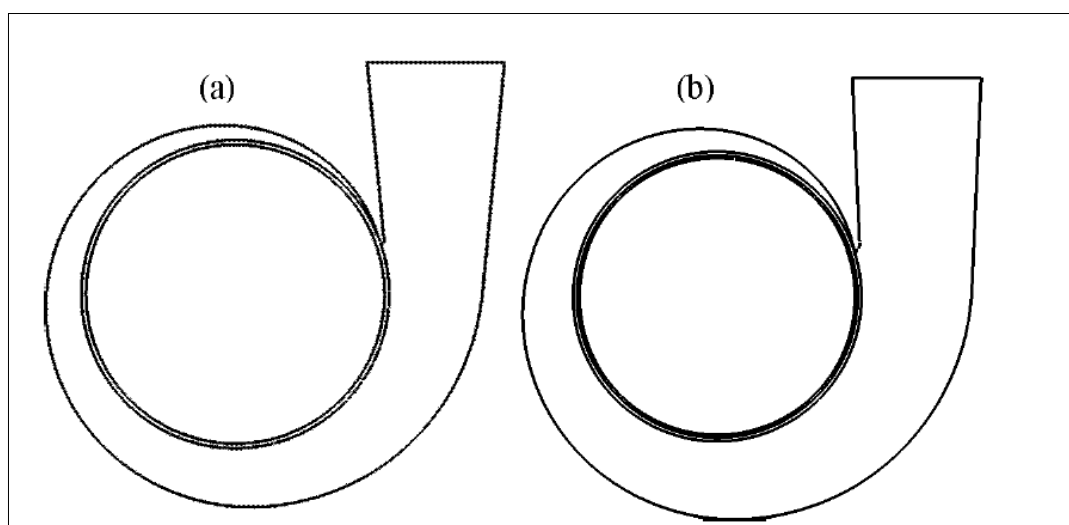


Computational fluid dynamics (CFD) is diligent technique which predicts the complex fluid flow in these types of machines. CFD uses the finite volume methodology, where the fluid domain is discretised into elements to numerically simulate the flow. The commercially available CFD software help to identify the flow regimes and eddies in the flow. It is possible to predict the flow and fatalities that occurs in the flow, which is cost effective methodology. The performance of the machine can be increased by parametrically varying various constraints.

## 2. Literature review

It is deduced that the flow in turbomachinery is associated with significant and accountable amount of losses. These fatalities lead to poor system efficiency. Many researchers have attempted to improve the performance of these devices using various techniques. Since CFD is a handy tool to simulate the flow characteristics, researchers have come up with various CFD methodologies for flow simulation. A glimpse of usage of CFD for turbomachinery applications is detailed as follows.

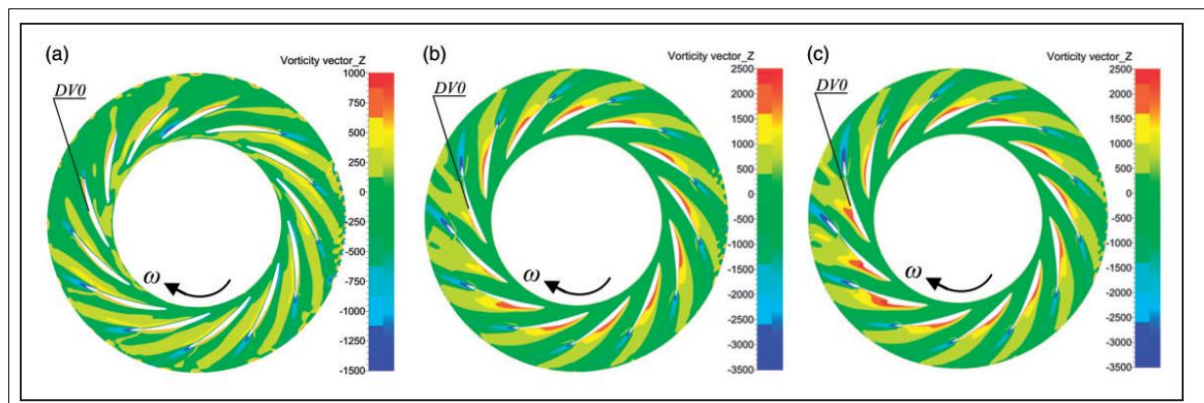
A numerical study was carried out by Alemi et al. [1] on a lower specific speed centrifugal pump. The analysis was performed for the designed and off-designed flow rate conditions. The pump geometrical dimensions were opted from Kelder et al. [2]. The validation was assessed with an already available experimental result. It was observed that the angular distribution of the static pressure around the impeller is nearly constant for the designed mass flow rate. The off-design operating conditions leads to noise and vibrations due to large radial fluid force from the impeller outflow and non uniform distribution of static pressure. The 3-Dimensional flow developed in the pump is very complex due to curvature profile of the blades and interaction of fluid in the pump components directing to flow separation. Calculations were performed at different flow rate ratios ( $\phi/\phi_n$ ) of 0.825, 1, 1.12, where  $\phi$  is fluid flow rate at off design conditions and  $\phi_n$  is designed fluid flow rate. The commercially accessible standard CFD code was used to solve Reynolds-averaged Navier–Stokes (RANS) equations. The numerical simulation was carried out using the three different turbulence models viz. standard  $k-\epsilon$ , the low-Re  $k-\omega$ , and shear stress transport (SST)  $k-\omega$  for the wide operating conditions, the  $k-\epsilon$  turbulence model is found to be more accurate at lower mass flowrates and  $k-\omega$  model is found to be more precise for the off-design operating conditions. The authors further considered two different volute designs viz. Pfleiderer design for constant angular momentum and constant tangential velocity with the Stepanoff design is as shown in figure 1.



**Figure 1.** 2-D Design of Volute (a) Pfleiderer design (b) Stepanoff design. [1]

They observed that Pfeleiderer volute design gave higher head and better efficiency at lower capacity and Stepanoff design at higher flowrate due to fewer radial forces within the volute. The study was further extended to the different rectangular, circular and trapezoidal volute cross sectional shapes having Stepanoff design for varying mass flowrates. The circular and trapezoidal shapes had intense radial forces at design operating point at the same time they gave better performance at higher flowrate. From the study it can be concluded that Stepanoff design with circular cross section volute shape is the optimised design.

The circumferential asymmetry of volute leads to the circumferential distortions in the flow. These distortions decay the aerodynamic performance of the machine. These unsteady pressure fields were studied by the Qi and Hong [3] for different fluid inlet conditions to verify the interface dynamic actions between volute and impeller. The authors performed the transient numerical flow simulation for the near stall, medium and higher flow rate conditions. CFD solver NUMECA was used to solve the 3-D RANS equations. The one-equation Spalart–Allmaras (SA) model without wall function was used as the turbulence model. The vorticity vector of the diffuser vanes for different mass flow rate is as shown in figure 2. The  $DV0$  is the location of the mid span of the diffuser.  $\phi$  is measured flow rate and  $\phi_n$  designed flow rate.

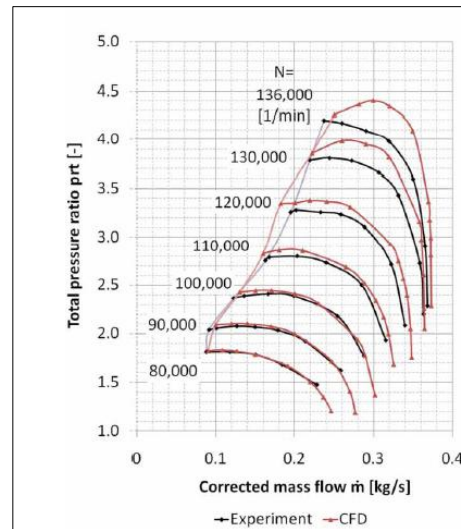


**Figure 2.** Time averaged vorticity fields in the diffuser vanes for  $\phi/\phi_n$  (a) 0.6 (b) 1.0 (c) 1.2. [3]

At higher flow rate the vorticity is generated on the pressure side of the vane because the flow at the inlet of the diffuser tend to move in the radial direction and has high velocity gradient. The medium flow condition also has the similar condition as experienced at the higher flow rate, but the influence is comparatively less. The pressure is more dominated at the suction side because the flow tends to move in the tangential direction causing the flow to deteriorate leading to comparatively better pressure recovery. It is also inferred that static pressure recovery coefficient and the total pressure loss coefficient at the diffuser flow channels are more influenced by the upstream flow at the volute tongue and less effected by the downstream for the middle and higher mass flow rate. The flow becomes unstable during near stall operating flow rate due to presence of flow fluctuations causing vibrations in the machine.

The transonic high-pressure ratio centrifugal compressor was investigated by Zheng et al. [4]. The 3-D flow is simulated using CFD technique for the designed pressure ratio of 4.2:1. The SA one - equation model was used as a turbulence model. The simulation method was performed using the frozen rotor technique. The aerodynamic performance of the volute was severely affected while operating at lower mass flow rate. The supersonic flow is expected at the impeller downstream while pressure ratio in the machine exceeds 4.5. At higher inlet velocities the range of operating conditions become narrower due to reduction of the incidence angle at stall. The volute acts as nozzle at higher flow rate and behaves like a diffuser at lower mass flow rate relative to designed flow. The numerical simulations were accomplished using the NUMECA solver based on finite volume scheme. The numerical and experimental result validation is as shown in figure 3, the plot represents the total pressure variation for different mass flow rates. To capture the shockwave and viscosity resolution the spatial discretization is

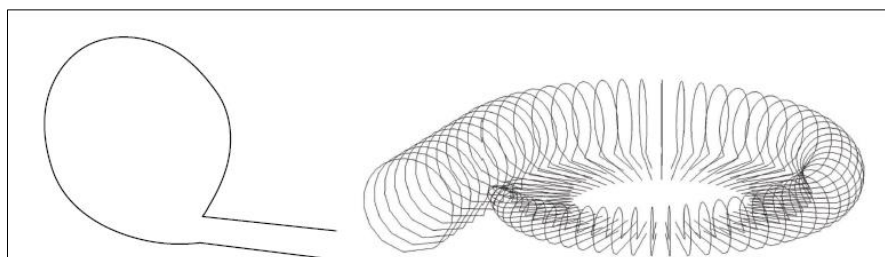
done using central difference scheme. The simulation is further performed with the  $k-\epsilon$  turbulence model to check the variation w.r.t previously used model at 130000 rpm and it is found to be in good agreement. The detailed study of the machine is performed with and without volute to check the major losses occurring. The swirl losses, frictional losses, expansion losses and mixing losses are present in the volute. It is identified that the swirl losses are



**Figure 3.** Validation of the CFD and experimental data. [4]

the major losses in the machine and is dependent on the radial velocity at the exit of the diffuser. Expansion losses are reducing up to the design point and later increases at the choking point. It is observed that there is 6.5% drop in the efficiency and 11.8% relative drop in the total pressure ratio due to volute influence.

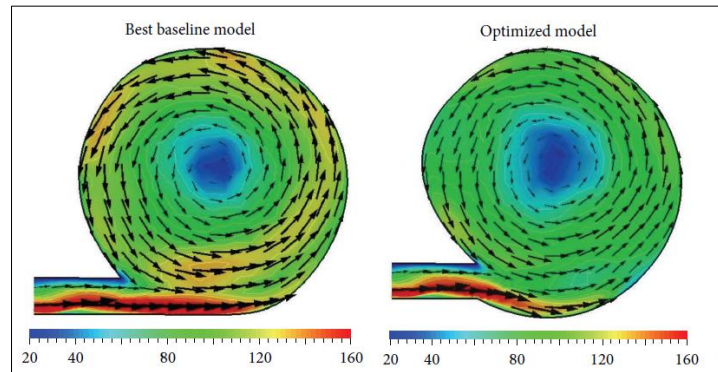
Heinrich and Schwarze [5] performed the numerical analysis on centrifugal compressor to obtain the optimised numerical model for the volute using the genetic algorithm. The cross section of the volute is represented as cubic B-splines. The control points are marked along these splines. The spline is duplicated and rotated along the axis as shown in figure 4. The various volute shapes were designed in order to optimise the model. The Genetic algorithm is executed for the different volute shapes with several iterations resulting in 1500 different shapes. The average isentropic efficiency was set as the main objective function of the volute. The volutes which gave the linear correlation between the isentropic efficiency versus average total pressure ratio graph plot were considered for further study by eliminating the other points.



**Figure 4.** Generated spline of the volute. [5]

The optimised model is further numerically simulated using the  $k-\omega$  turbulence model. The optimised model having base meridional velocity magnitude is as shown in figure 5. It is observed that the velocity gradients are reduced near the wall comparatively base model for the optimised model. The reduced velocity gradients lessen the turbulence intensity and reduce frictional losses. The static pressure

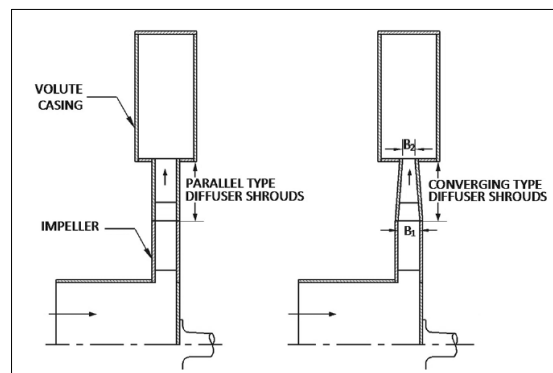
recovery was increased by 32% at off-design condition along with reduction in total pressure loss by 12%. It is observed that the flow velocity reduced in the optimised model at the cost of the increases static pressure.



**Figure 5.** Meridional velocity Magnitude (m/s). [5]

The optimised volute was having the sharp corner at the diffuser exit and inner wall of volute, this reduces the generation of the secondary vortex development in the flow. The curved wall near the volute results in a deflection of the jet.

Madhwesh et al. [6] performed numerical investigations on a centrifugal fan for its improved performance by modifying the shrouds of vaned diffuser as shown in figure 6. They parametrically reduced the width of the diffuser at its exit in terms of convergence ratio (CR). The  $k-\epsilon$  turbulence model available in ANSYS Fluent was used to capture the turbulent characteristics. They found that on reducing the width of the diffuser, fan performance becomes better up to an optimum convergence ratio due to streamlining of the flow in the diffuser passages.



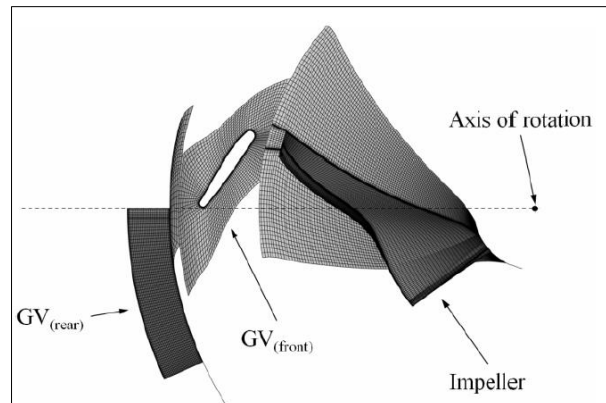
**Figure 6.** Meridional view of the modified geometry.[6]

It is revealed from this study that for the optimum value of the convergence ratio of 0.35 an overall improvement of about 3.6% is observed for head co-efficient and 2.1% for theoretical efficiency. They also justified that the configurations with CR beyond 0.35 become redundant since a jet-like flow in the diffuser passages enhances turbulence generation.

González and Santolaria [7] obtained the numerical relation between the dynamic flow structure and global variables for a centrifugal pump having lower specific speed. A sliding mesh technique was used to simulate the interaction between the impeller and volute. They observed that the pressure field on the blades give rise to loading which in turn leads to the torque. A strong eddy flow is identified at the exit of the impeller for higher flow rate which leads to depreciation in pressure recovery. The authors showed that there is an improved mixing process during off-design operation.



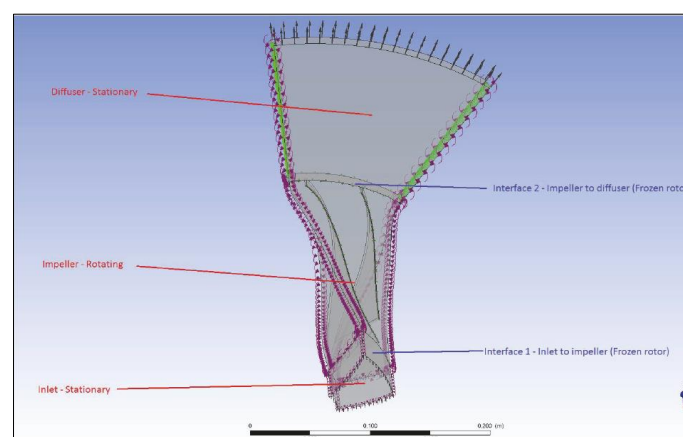
Jung et al. [8] carried out the numerical and experimental study on the centrifugal fan to maximise its overall performance. The six design parameters were selected to get optimum design. The CFD analysis was done using the ANSYS Turbo-grid and ANSYS ICEM. The 3-D RANS equations were solved with the help of the commercially available CFX code. The discretised computational domain is as shown in figure 7.



**Figure 7.** Computational domain of the impeller with design parameters. [8]

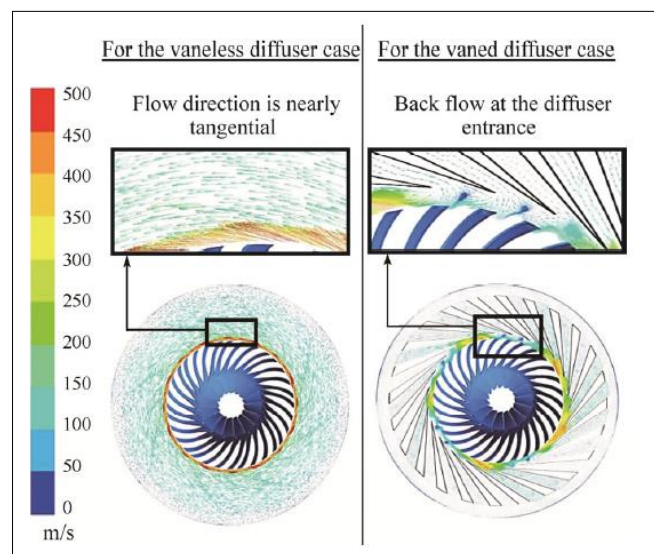
The turbulence model SST k- $\omega$  was used for numerical simulation. The various dimensions were varied parametrically to get the optimum design. The simulation results were validated with the experimental values. The Beta angle of trailing edge (TE) of the rear guide vane (GV) has its influence on the performance along with the number of GVs and thickness of the rear GV. It is observed that loss in pressure is due to the decrease in the cross-sectional area between the guide vanes which leads to generation of the vortex near the hub.

Dewar et al. [9] numerically and experimentally analysed single blade passage of the centrifugal compressor. The static pressure calculation is obtained by the experimental and CFD method. The computational domain is as shown in figure 8. Three mass flow rates were selected for three different rotational speeds. The 12 location points were selected on the diffuser to measure the static pressure. The simulation was performed using frozen rotor procedure to study the interaction between the diffuser and impeller. ANSYS CFX is used for mathematical simulation. The SST k- $\omega$  turbulence model was used for the numerical simulation. The single blade passage was designed with an assumption of circumferential uniformity. The optimal interface of impeller and diffuser position is found to be 2% of the blade radius. It is observed that the optimum distance gives an improved performance up to 10% of the diffuser radius and at greater radius it is very less influential.



**Figure 8.** Blade passage along with interface zones. [9]

Halawa et al. [10] investigated the influence of the rotating stall in the vaned and vaneless diffuser in the centrifugal compressor. Preliminary analysis was carried for vaneless diffuser and then for vaned diffuser. The reverse flow in the diffuser was studied when the surging occurs in the machine. The flow governing equations were solved using the ANSYS Fluent. The k- $\epsilon$  realizable turbulence model was used. The transient simulation was performed for the transonic flow at higher speed using sliding mesh technique. It is observed that the pressure fluctuations are more at the entrance of the vaneless diffuser when compared with vaned diffuser at the surge point. The authors also depicted that development of the surge occurs more in the case of the vaned diffuser. The flow is tangential with lower velocity at the exit of the vaneless diffuser. The backflow is observed at the exit of impeller and the stalling is detected at the entry of the vaned diffuser as shown in the figure 9.



**Figure 9.** Velocity vector plots at the surge point. [10]

The study shows that the stall region is similar in the impeller for the vaned diffuser and for the same condition it varies from blade passage to passage for the vaned diffuser. The vaneless diffuser gives better performance when related to vaned diffuser during surging because of the constant distribution of the pressure on blades.

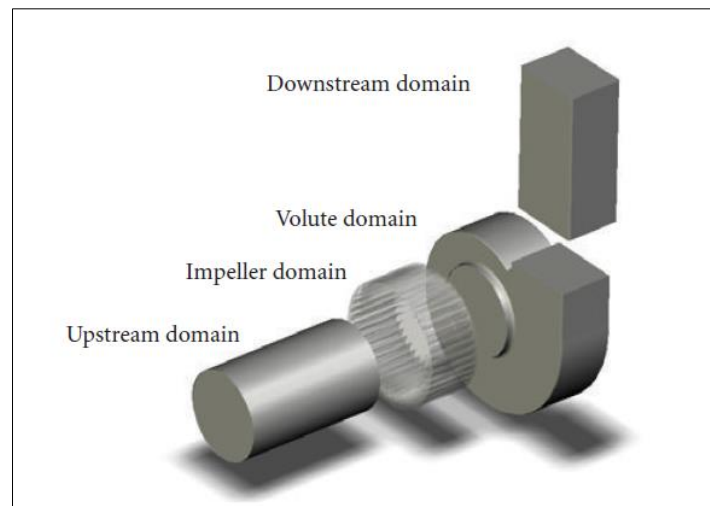
An aerodynamic analysis for industrial centrifugal blower was carried out by Hariharan and Govardhan [11]. The three different volute shapes viz. parallel wall, rectangular and circular shapes with two area ratios [ARs] 4 and 5 were used for the numerical analysis. The area ratio is defined as the ratio of the volute width to the width of impeller. The performance of these volutes is numerically evaluated with the help of finite volume method [FVM] using commercially available CFD software ANSYS CFX. A 2-equation turbulence model k- $\epsilon$  is used. The blower was tested for design and off-design condition. The volute shapes of rectangular and parallel wall with AR 4 and circular cross section with AR 5 gave better performance. The authors investigation concluded that parallel wall volute is more efficient and compact.

Madhwesh et al. [12] performed numerical study on the performance of the backward swept centrifugal fan. The fan was designed with circular shroud fences. These fences were varied parametrically to evaluate the performance of the centrifugal fan. The numerical simulation was carried out with the CFD approach. The k- $\epsilon$  turbulent model was used for simulation. The relative flow in the system was executed using the sliding mesh technique. It is found that fence on the shroud surface of impeller have substantial influence on the performance of the fan. The modified model with the optimised fence diameter on the shroud gave 2% increase in the head coefficient along with 2.3% increase in the efficiency when associated with a base model. The shroud fence on the front face was

comparatively not more influential on the performance of the centrifugal fan. Providing fence close to the tip of the impeller on back shroud will give a better performance with higher static pressure and efficiency.

Karant and Sharma [13] analysed the fluid flow characteristics at the radial gap between the diffuser entry and impeller exit. A judicious solution has been deduced using the numerical techniques based on CFD calculations. The geometrical configuration was modified by varying the radial gap between the impeller and diffuser. The outer radius of the impeller and diffuser is kept constant, the inlet radius of the impeller is varied to get the radial gap ratio. The flow is very multifaceted in the radial gap because of the formation of the recirculation zones leading to intensification of pressure losses. The increased recirculation zones lead to stalling of the blades therefore, these wake regions must be abridged. The optimum radial gap will provide a better performance with minor loss coefficient. The jet wakes in the flow were also influenced with the modified design. The better dynamic head and static heads were developed in the fan for an optimum radial gap ratio of 0.15.

Younsi et al. [14] evaluated the performance of the forward swept centrifugal fan during the unsteady flow. The geometric model of the centrifugal fan is as shown in figure 10. The numerical and experimental analysis is performed for four different configurations to study unsteady flow behaviour.



**Figure 10.** Geometric model of the forward curved centrifugal fan. [14]

The volute casing dimension is kept constant by varying other parameters like number of blades, blade spacing, volute tongue and radial distance between impeller edges. The aerodynamic characteristics is evaluated with CFD to solve the RANS equations, SST  $k-\omega$  turbulence model is used for analysis. The study shows that irregular spacing of the blades had less influence on the aerodynamic characteristics when compared with base model. The impeller with reduced outlet diameter is more efficient because it requires less torque, however pressure drop is compensated by increasing the impeller speed. The impeller with minimum number of blades is more resourceful than base model due to reduced aerodynamic losses.

Pettit and Nilson [15] compared the steady and unsteady simulation of the centrifugal pump with vaned diffuser. The computational domain consisted unshrouded impeller with 12 vaned diffuser. The results of the steady and unsteady were validated with the already available experimental data. Frozen rotor technique is used for steady state simulation. The sliding grid technique is implemented for the unsteady technique, the impeller is rotated adjacent to diffuser. The different turbulence models were used for validation viz.  $k-\epsilon$ , realizable  $k-\epsilon$ , RNG  $k-\epsilon$  and SST  $k-\omega$ . The  $k-\epsilon$  turbulence model is found to predict more accurate comparatively to other models. The numerical results at the diffuser passage did not match with the experimental values during steady state simulation. The unsteady simulation



predicted nearly correct values with the experimental values. It was observed that the jet wakes at the impeller exit are diffused at the exit of the diffuser during unsteady simulation.

### 3. Conclusions

This article highlights the usage of CFD for turbomachinery applications. Following are the main highlights of this study.

- CFD is very useful in simulating the flow in turbines, compressors and centrifugal pumps, thus identifies core region of interest where flow losses are dominant.
- Various methodologies involved in CFD analysis are dealt in detail.

Finally, it is concluded from the article that CFD tools can be used to re-design the turbomachinery components for possible performance augmentation along with detailed capture of flow physics and flow visualization.

### References

- [1] Alemi H, Nourbakhsh S A, Raisee M and Najafi A F 2015 Effects of volute curvature on performance of a low specific-speed centrifugal pump at design and off-design conditions *J. Turbomach.* **137** 1–10
- [2] Kelder J D H, Dijkers R J H, Van Esch B P M and Kruijff N P 2001 Experimental and theoretical study of the flow in the volute of a low specific-speed pump *Fluid Dyn. Res.* **28** 267–80
- [3] Qi G W and Hong Z X 2016 Effect of a volute on the unsteady flow in the vane diffuser of a centrifugal compressor *Proc. Inst. Mech. Eng. Part A J. Power Energy* **230** 773–91
- [4] Zheng X Q, Huenteler J, Yang M Y, Zhang Y J and Bamba T 2010 Influence of the volute on the flow in a centrifugal compressor of a high-pressure ratio turbocharger *Proc. Inst. Mech. Eng. Part A J. Power Energy* **224** 1157–69
- [5] Heinrich M and Schwarze R 2017 Genetic optimization of the volute of a centrifugal compressor *12th Eur. Conf. Turbomach. Fluid Dyn. Thermodyn. ETC 2017* 2016
- [6] Madhwesh N, Vasudeva Karanth K and Yagnesh Sharma N 2018 Investigations into the Flow Behavior in a Nonparallel Shrouded Diffuser of a Centrifugal Fan for Augmented Performance *J. Fluids Eng. Trans. ASME* **140**
- [7] González J and Santolaria C 2006 Unsteady flow structure and global variables in a centrifugal pump *J. Fluids Eng. Trans. ASME* **128** 937–46
- [8] Jung U H, Kim J H, Kim J H, Park C H, Jun S O and Choi Y S 2016 Optimum design of diffuser in a small high-speed centrifugal fan using CFD & DOE *J. Mech. Sci. Technol.* **30** 1171–84
- [9] Dewar B, Tiainen J, Jaatinen-Värri A, Creamer M, Dotcheva M, Radulovic J and Buick J M 2019 CFD Modelling of a Centrifugal Compressor with Experimental Validation through Radial Diffuser Static Pressure Measurement *Int. J. Rotating Mach.* 2019
- [10] Halawa T, Alqaradawi M, Gadala M S, Shahin I and Badr O 2015 Numerical investigation of rotating stall in centrifugal compressor with vaned and vaneless diffuser *J. Therm. Sci.* **24** 323–33
- [11] Hariharan C and Govardhan M 2019 Aerodynamic performance and flow characteristics of an industrial centrifugal blower volute with varied cross-sectional shapes/area ratios *Int. J. Turbo Jet Engines* **36** 89–106
- [12] Madhwesh N, Vasudeva Karanth K and Yagnesh Sharma N 2018 Effect of innovative circular shroud fences on a centrifugal fan for augmented performance - A numerical analysis *J. Mech. Sci. Technol.* **32** 185–97
- [13] Karanth K V and Sharma N Y 2009 CFD analysis on the effect of radial gap on impeller-diffuser flow interaction as well as on the flow characteristics of a centrifugal fan *Int. J. Rotating Mach.* 2009 1–8
- [14] Younsi M, Bakir F, Kouidri S and Rey R 2007 Influence of impeller geometry on the unsteady flow in a centrifugal fan: Numerical and experimental analyses *Int. J. Rotating Mach.* 2007
- [15] Petit O and Nilsson H 2013 Numerical investigations of unsteady flow in a centrifugal pump with a vaned diffuser *Int. J. Rotating Mach.* 2013.