Journal of Advanced Research in Instrumentation and Control Engineering
Volume 8, Issue 1&2 - 2021, Pg. No. 1-5
Peer Reviewed Journal

Research Article

# CFD Analysis of Centrifugal Air Compressor

Putta Vithal Naga Kumar[1], VV Prathibha Bharathi[2]

[1]Research Scholar, Nova's Institute of Technology, Eluru, Andhra Pradesh, India.
[2]Head of R&D, Formulate Tech Private Limited, Hyderabad, Telangana, India.
DOI: https://doi.org/10.24321/2456.1398.202101

## I N F O

**Corresponding Author:**
Putta Vithal Naga Kumar, Research Scholar, Nova's Institute of Technology, Eluru, Andhra Pradesh, India.
**E-mail Id:**
puttavithal@gmail.com
**Orcid Id:**

**How to cite this article:**
Kumar PVN, Bharathi VVP. CFD Analysis of Centrifugal Air Compressor. *J Adv Res Instru Control Engg* 2021; 8(1&2): 1-5.

Date of Submission: 2021-04-14
Date of Acceptance: 2021-05-11

## A B S T R A C T

A single-stage industrial centrifugal compressor is designed to operate at low- to go against the flow. An analysis technique for impeller design, incorporating an aerodynamic constraint is defined that involves solving for a value known as "t" and a suction ratio. One dimension of the impeller is constructed with radial tips and the other dimension uses vanes to help with diffusing the flow. In this paper, the theoretical calculation is only in terms of efficiency, the original ones were measured where the geometry modelling and mesh development is performed in the Gambit Pre-processor. Computational dynamic two-dimensional study of flow in the flow passage and pressure units of an in an impeller; and its interaction with the volute are done in FLUENT and then CFD results are compared to the theoretical results.

**Keywords:** GAMBIT, FLUENT, CFD, Volute, Vanes

## Introduction

A "dynamic centrifugal compressor" is one which rotates independently to produce compression and expansion. This is a form of centrifugal compressor, also known as a centrifugal impeller or radial compressor, where the compressor shaft is located on an axle that houses a variety of impeller-shaped blades. A centrifugal compressor works by using the power of mass air's centrifugal force to attain its own force of compression. An efficient and dynamic fluid-expansion rotary compressor raises the pressure by allowing for continuous flow of kinetic energy and velocity through the rotor or impeller. This happens with the centrifugal pump, as well: the pump impeller pushes against the side of the inlet, but also rotates outwards. In the case, static pressure decreases as the centrifugal force drops but increases as well because of the decrease in relative velocity. It is proposed that the results of the present study include designing a typical industrial centrifugal compressor. When Aung, the creator of the design, begins the process of expanding, all claims must be heard and treated equally. One-dimensional performance analysis will have to be done before the design is completed and a second iteration is undertaken. The full design of the gas path must be investigated for the entire geometric shapes of the hub and shroud. Finally, FLUENT's 2-blade geometry has been successfully applied to a 3-D flow analysis in the form of the extracted GAMBIT.

## Literature Survey

A suitable numerical design is found for the centrifugal compressor which is demonstrated for a constant isentropic Mach number along the blade suction and pressure sides. Velocity is calculated for each of a specific impeller geometry, and further analysis is done on the same data obtained by the 3D analysis programme. The desired pressure and flow mean and actual geometry mean velocities are used to calculate the overall mean speeds and swirls that are applied to the fluid. The expansion and cooling ends are organised to give the flow a prescribed velocity throughout the interior of the casing, regardless of the rotation speeds of the hub and rotating machinery. The blade shape is expanded until a prescribed by the required amount of swirl is obtained. This procedure has been applied to the design of centrifugal compressor to provide an example of that helps to illustrate the principles of working them into practise.[1]

The efficiency and flow measurement study in an aerodynamic impeller ratio of 4:1 pressure flow structure was synthesised on the premise of viscous CFD results. A successful CFD calibration was done using a laser interferometry probe was made to balance measurements made with pneumatic methods. The CFD model solidified, taking on a major responsibility for being a dependable representation of the impeller internal flow system and getting Performance incorporated into it. The results shown in the section also indicate the loss generation and secondary structure of the impeller. Conclusions drawn from this show that while the overall impeller efficiency is high, the recovery potential in the diffuser is underutilised, which leads to performance degradation for the diffuser as crossflow. A parametric design may be needed to doer could call for an expansion A procedure for aerodynamic performance is laid out and the potential gains in overall diffusivity that could be realised by changes to the relative diffusion are estimated.[2]

The impact of impeller and diffuser effects on a centrifugal compressor stage is interesting to a designer to measure, whether it is steady or unsteady the optimization of a particular components. The sustained CFD calculations on transonic centrifugal stage compressors can be found in this paper. Stable flow has shown to be capable of making excellent predictions about the diffuser and pump output as well as this stage's, particularly when there is a stationary mixing plane underneath it. Flow phenomena highlighted by computational analysis to be important in the design of a centrifuge speed lines are viewed as an extend of line comparison to real world lines from tests to display.[3]

Scientists predict that a new 3D technology from NASA would offer precise, efficient, and predictable wind power generation in a totally stable manner. According to this, present measurement, it has high tip-sephemer flow. The cause of this can be found in the difference between the tiphiids and the model for the blend-dye flow, though. the measurement results in a considerable pressure loss on the shroud/ in the quadrant closest to the exit of the impeller. In, addition, the map further predicts a zone of meridional backflow just below the shroud. The latest ideas for future flow research in NASA impellers have been put forward expanded, something inside.[4]

Without the volute, the centrifugal engine, the volutes will lose the capability to expand. The nature of the volute has both a big impact on the compressor performance, as well as on the operating range. This 3D flow simulation was created to comprehend the flow mechanisms and to assist in the design of volute flow system design to meet performance requirements. The flow within the V-Navier flow cell is modelled using the Navier-Stokes equations for the vaneless diffuser/ pyriform equations, a type of diffuser where the flow is assumed to be viscous. The complex flow structures for different volar geometries are simulated for various considerations, including vortex stability and geometrical correction. The data used in the calculations lines up with the results from the experiments, which are both accurate and satisfying. Results also include more information on the ways to go in determining the shape of a volute.[5]

## Methodology

The design of a centrifugal compressor is begun with clear design objectives in mind and the use of components that have been determined to work towards these goals. A less resource-intensive preliminary design would help guide to project development will produce the candidates with the least number of inputs. However, it must be capable of responding to a variety of circumstances, as well as to those found in the design process of design. This system should come up with a design for the entire stage that's being built and then optimise it, checking each of the stages and their components are aligned. The best practises in designing a preliminary design must support the planned method of conducting the design/weaknesses and applying those practises to refine the design are unavoidable Until aerodynamic evaluations and calculations are used, it provides an initial approximation of each component's geometry. Afterwards, it goes into additional aerodynamic design for further study and improvement. (Figure 1).

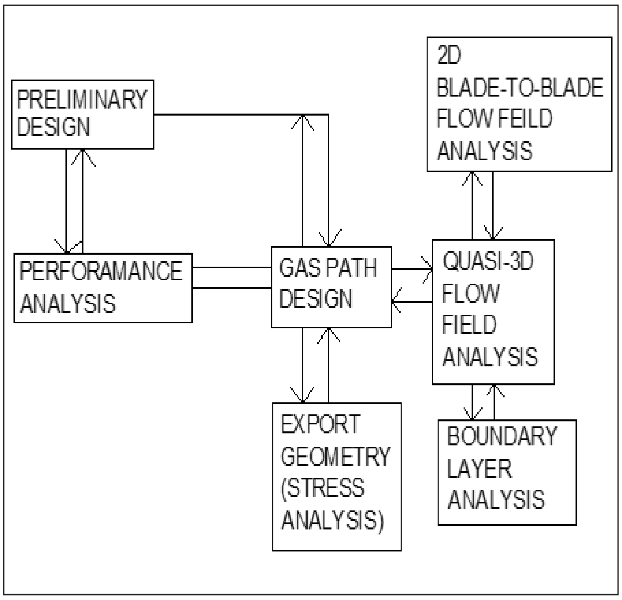


**Figure 1.Detailed design flow chart**

## Design Calculation

"Design and analysis of a single stage; radial flow – centrifugal air compressor with a suction pressure of 1bar and volume flow rate of – 50,000. The in let flow is assumed to be axial, and the compressor's impeller is designed for the following constraints:

- Stage flow co-efficient $\varphi$ = 0.14
- Impeller rotational speed, $N$ = 15,000 rpm
- Stage efficiency, $\eta_{st}$ = 80%
- Slip factor, $\sigma$ = 0.88
- Working fluid – Air
- Suction pressure – 1bar
- Flow – 500,000 $NM^3$/sec
  = 13.88$m^3$/sec

## Impeller Calculations

**Stage flow co-efficient $\varphi$ = 0.14; N = 15,000rpm**

$$\frac{m}{\pi\rho_0 r_2^2 v_2} = \frac{Q_o}{\pi r_2^2 v_2} \quad \text{AND} \quad U_2 = \frac{\pi r_2 N}{30}$$

$$\Rightarrow r_2 = 0.27m \;\; ; \; U_2 = 424.1\,m/sec$$

**Assume $r_1$= 0.165m**

$$\rho_0 = \rho_1 = \frac{p_{01}}{RT_{01}} = 1.276kg/m^3$$

$$m = \frac{Q_0}{\rho_0} = 17.71\,kg/sec$$

$$C_1 = \frac{m}{\rho_1 A_1} = 162\,m/sec$$

$$U_1 = \frac{\pi r_1 N}{30} = 259.18m/s$$

$$T_1 = T_{01} - C_1^2/2c_p = 260°$$

$$p_1 = p_{01}(T_1/T_{01})^{\gamma/\gamma-1} = \overline{0.843}$$

**Stage pressure ratio:**

$$P_{ro} = P_{04}/P_{01} = \{1 + \eta_{st}/c_pT_{01}(\sigma v_2^2 - v_1^2)\}^{\gamma/\gamma-1} = \overline{2.365}$$

$$T_2/T_{01} = 1 + U_2^2/2CpT_{01} = 1.327$$

$$T_2 = 362°$$

$$p_2/p_{01} = (T_2/T_{01})^{\gamma/\gamma-1}$$

$$p_2 = \overline{2.69}$$

$$\beta_1 = tan^{-1}(C_1/U_1) = 32°$$

## Number of impeller blades

**Balje's formula:**

$\sigma = \{1+ 6.2/Zn^{2/3}\}^{-1}$

n = impeller tip dia/ eye tip dia

**Volute:**

**Stanatz's method:**

$\sigma = 1 - 1.98/z \Rightarrow z = 16.5 \approx 17$

**Stodal's theory:**

$\sigma = 1- \pi/z \Rightarrow z = 26.179 \approx 27$

Here, 27 blades are considered for the design and analysis.

## Vaneless Diffuser

**For vane less diffuser**

$tan\ \alpha_2 = 0.26+3\varphi$

$\alpha_2 = 34.2°$

$\alpha_4 = 30° + (\varphi/(0.06)^2 = 35.4°$

$r_4 = (1.55 + \varphi)r_2 = 456.3m$

**Volute**

$$rc_\theta = r_2c_{\theta 2} = r_3c_{\theta 3} = K = 211.13m^2/s$$

$$\frac{Q}{Kb_3} = 0.365$$

$$r_4 = r_3\,exp\left(\frac{\theta}{360}\frac{Q}{Kb_3}\right)$$

$$\Rightarrow r_4 = 479.69mm, \text{At} \quad \theta = {}^{\pi}/_{4}$$

## Result

A single stage, radial flow-centrifugal air compressor is designed with the required parameters and suitability assumed constraints. The following dimensional and flow results are obtained from the prescribed calculations.

## Impeller

- Impeller eye diameter: 330mm
- Impeller tip diameter: 540mm
- Impeller blade root angle $\beta_1$: 32°
- Impeller flow inlet angle $\alpha_1$: 90°
- Impeller blade tip angle $\beta_2$: 90°
- Impeller tip flow angle $\alpha_1$: 34.2°
- Absolute flow at inlet $U_1$: 259.18*m/s*
- Absolute flow at exit $U_2$: 424.1*m/s*
- Suction pressure $P_1 = \overline{0.844}$
- Discharge pressure $P_2 = \overline{2.69}$

## Diffuser

Diffuser exit diameter: 456.3mm

**Table 1.Volute data**

| $\theta$ | $\frac{\pi}{4}$ | $\frac{\pi}{2}$ | $\frac{3\pi}{4}$ | $\pi$ | $\frac{5\pi}{4}$ | $\frac{3\pi}{2}$ | $\frac{7\pi}{4}$ | $2\pi$ |
|---|---|---|---|---|---|---|---|---|
| $r_4$(mm) | 479.69 | 504.28 | 530.14 | 557.3 | 585.9 | 615.9 | 647.53 | 680.7 |

ISSN: 2456-1398
https://doi.org/10.24321/2456.1398.202101

Compressors exit pressure $P_4 = \overline{2.32}$

## I-D Aerodynamic Analysis

One-dimensional flow analysis involves many features common to the different types of components to be analysed. It is convenient to cover these common features first. The basic solution is accomplished on a mean stream surface, subject to appropriate boundary conditions and conservation equations and supported by suitable empirical models.

**Torque:**

$$\tau = m[r_2 C_{U2} - r_1 C_{U1}] = 2212.5$$

**Rise in Total Enthalpy:**

$$h_{t2} - h_{t1} = (U_2^2 - U_1^2)/2 = 56.34kj$$

**Blade Loading Loss:**

$$\overline{w}_{BL} = (\Delta W / W_1)^2 / 24 = 0.86$$

**Wake Mixing Loss:**

$$\overline{w}_{mix} = [(C_{m,wake} - C_{m,mix})/W_1]^2 = 0.644$$

## Discussion

Figure 2, shows the post processing window where we can see the final results of the analysis process. In this we can see the contours of the pressure, velocity, temperature, wall fluxes, density etc.

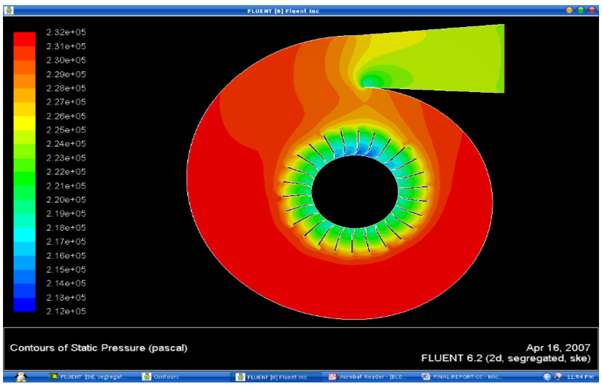


**Figure 2.Contours of Static Pressure**

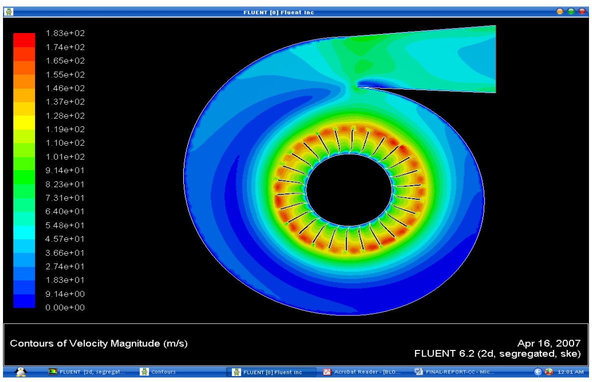


**Figure 3.Contours of Velocity Magnitude**

The above Figure 3 shows the CONTOURS OF STATIC PRESSURE of the analysis. Here the pressure variations are shown by different colours. In this the maximum pressure is shown with the red colour, the maximum pressure obtained is $\overline{2.15}$. And the minimum pressure of this analysis is $\overline{2.32}$. In post processor we can also plot contours for dynamic pressure, absolute pressure and for the total pressure.

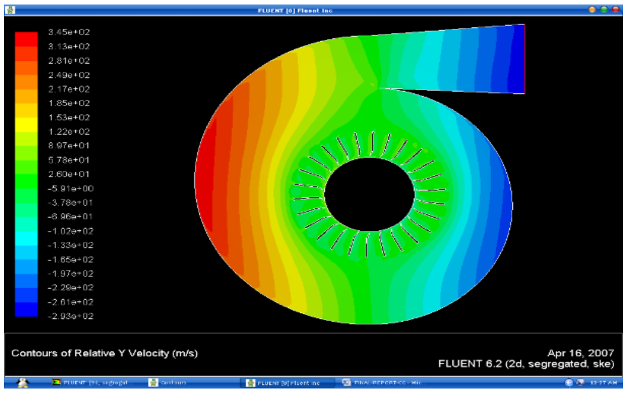


**Figure 4.Contours of X-Velocity and Y-Velocity**

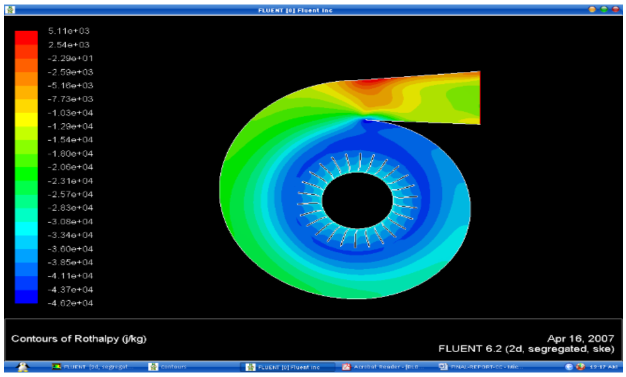


**Figure 5.Contours of Rothalpy - Temperature**

Figure 4, Shows the contours of VELOCITY MAGNITUDE. And the maximum velocity obtained is 165 m/sec. and the minimum velocity magnitude obtained is 9.14m/sec. we can also see the plot results for the x-velocity, y-velocity in the Figure 5. In this post processing we can also see the contours of the other variables like turbulence, wall fluxes, density etc.

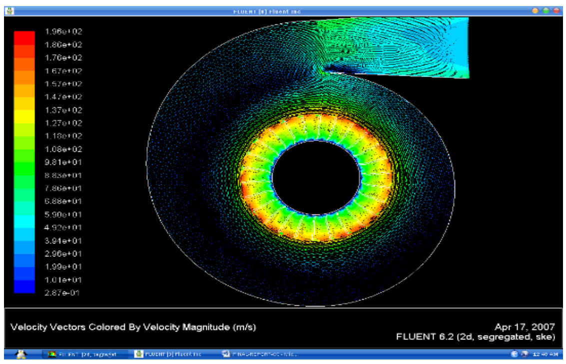


**Figure 6.Velocity Vectors Coloured by Velocity Magnitude**

ISSN: 2456-1398
https://doi.org/10.24321/2456.1398.202101

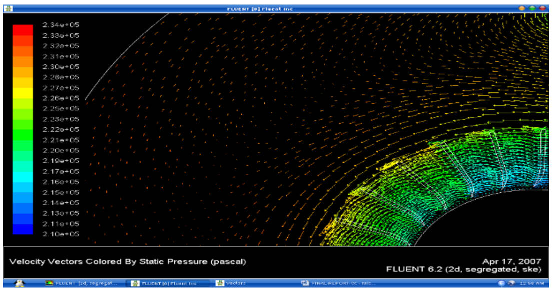


**Figure 7.Velocity Vectors Coloured by Static Pressure**

## Conclusion

The aerodynamic methodology for the obtainment of the geometry for the required flow rate and suction pressure is presented. Exit conditions for the given inlet requirements and the obtained dimensions are calculated for the two-dimensional analysis of complete compressor stage and the blade-to-blade impeller region. The results section shows different contours of pressure, temperature (rothalpy), velocity. Vectors of fluid flow velocity are also shown in every stage of the compressor.

Pressure increase is obtained from the theoretical calculations in the impeller and diffuser section. Slight pressure drop is attained in the volute stage, also from the calculations. This is supported in the analytical CFD results of the complete compressor stage. In the pressure contours of the compressor, it is clearly evident that there is pressure increase in impeller, diffuser and pressure drop in the volute section. Velocity vectors shown clearly state the flow direction at different stages of the compressor

## References

1. Pabrucker H, Braembussche RAVD. Inverse design of centrifugal impellers by simultaneous modification of blade shape and meridional contour. ASME. 200-GT-457.
2. Larosiliere LM, Skoch GJ. Aerodynamic Synthesis of a Centrifugal Impeller Using CFD and Measurements. NASA Technical Memorandum. 107515.
3. Srivastava BN, Macrorie M. Stage Analysis of Centrifugal Compressors for Design: CFD Validation. AIAA. 2004-4835
4. Moor J, Moore JG. A Prediction of 3-D Viscous Flow and Performance of the NASA Low-speed centrifugal compressor. NASA turbo machinery research group. Vol. Grant no. NAG. 3-919.
5. Cheng Xu AND Michael Muller. Development and design of a centrifugal compressor volute. *International journal of design rotating machinery* 2005; 3: 190-196.
6. Aungier RH. A textbook on “centrifugal compressors. [strategy for aerodynamic design and analysis] ASME press.
7. A Textbook of turbines, compressors and fans. S.M yahya. TATA Mc. GRAW HILL. Second edition.

ISSN: 2456-1398
https://doi.org/10.24321/2456.1398.202101