Comparison of operational effectiveness of a turbocharger volute.

ZALA, K., ASIM, T., BAKER, P., UBBI, K. and MISHRA, R.

2019

Comparison of operational effectiveness of a turbocharger volute

Karina Zala
University of Huddersfield
Queensgate
Huddersfield
+44 1484 472364
k.zala@hud.ac.uk

Taimoor Asim
Robert Gordon University
Aberdeen
+44 1224 262457
t.asim@rgu.ac.uk

Paul Baker
University of Huddersfield
Queensgate
Huddersfield
+44 1484 472979
p.c.baker@hud.ac.uk

Kuldip Ubbi
University of Huddersfield
Queensgate
Huddersfield
+44 1484 472979
k.s.ubbi@hud.ac.uk

Rakesh Mishra
University of Huddersfield
Queensgate
Huddersfield
+44 1484 473263
r.mishra@hud.ac.uk

ABSTRACT

Centrifugal compressors are commonly used across a wide range of applications such as in the automotive industry for engine turbocharging. A turbocharger has four main components i.e. inducer, impeller, diffuser and volute. Turbocharger volutes are commonly designed by neglecting the effects of friction, however, in the real-world, frictional effects have a significant influence on the performance and efficiency of the volute and the turbocharger. This study focuses on the operational effectiveness of the turbocharger volute, making use of two asymmetric type volute models that have been designed for high-pressure centrifugal compressors. For this purpose, advanced Computational Fluid Dynamics (CFD) based techniques have been employed. Three dimensional models of the turbocharger compressor stage have been developed and analysed by monitoring the pressure fluctuations through the volutes. Incorporating frictional effects has been noticed to have prominent influence downstream of the tongue, in the tail of the turbocharger compressor volute. This study shows that designing the turbocharger volute, by overlooking the frictional effects, results in an overestimation of the pressure fluctuations observed within the volute. Therefore, from an operations perspective, it is beneficial to design the volute incorporating frictional effects for high-pressure centrifugal compressor applications.

Keywords

Computational Fluid Dynamics (CFD), Turbocharger, Centrifugal Compressor, Volute.

1. INTRODUCTION

Centrifugal compressors are universally used across many applications, such as turbocharging an engine. One of the main components of the turbocharger is the volute. In general, the volute for centrifugal compressor applications are in the same manner as that designed for fans and pumps, by neglecting the effects of friction, however, in reality, frictional effects play an essential role in the performance and efficiency of the volute and the turbocharger. Many researchers have explored the pressure fluctuations inside the volute. Parrondo-Gayo et al. used experimentation to study the effect operating points have on the pressure fluctuations at the blade passing frequency in the volute of a centrifugal pump. A single operational speed across various flow rates had been investigated and the findings state that the tongue has a significant influence on the interaction between the impeller and the volute as well as on the dynamic pressure generation in the volute and noise generation at off-design conditions [1]. Ballesteros-Tajadura conducted experimental and numerical investigations of pressure fluctuations in the volute of a centrifugal fan. Both methodologies were in good agreement with each other and the pressure fluctuations in the volute had been discovered to be a result of the aerodynamic field, which depicted the effects of jet-wake phenomena and the interaction of the blade-tongue [2]. Kaupert and Staubli used experimentation to study the unsteady pressure field in a high specific speed centrifugal pump impeller, focusing predominantly on the influence of the volute. The findings conclude that the unsteady variations in circumferential pressure are predominantly a result of the tongue behaving as a boundary that separates two regimes of flow [3]. These studies have been carried out on incompressible flow applications such as fans and pumps however, little research has been carried out for compressible flows in this regard, where the effects of friction plays an essential role on the operational effectiveness of the volute, for applications such as turbocharging. For this reason, the presented study focuses on numerically analysing the pressure gradients during the interactions between the impeller and volutes designed by (a) neglecting friction and (b) incorporating friction.

2. NUMERICAL SETUP

As part of the numerical investigation, a commercial code FLUENT® has been used to carry out instantaneous simulations. The geometry of the centrifugal compressor stage is shown in Figure 1. An extension at the inlet duct section, upstream the compressor stage and the outlet duct section, has been implemented. This has been defined by $3 \cdot D_{out}$ in order to achieve fully developed flow with minimal flow instabilities incurred in the compressor stage.

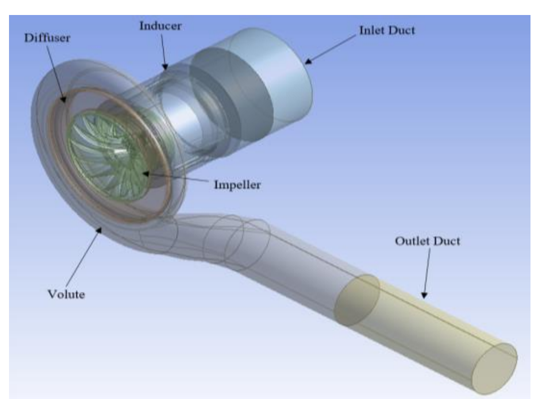
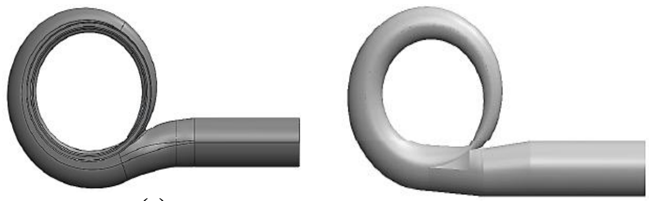


Figure 1. Geometry of the centrifugal compressor stage

Two asymmetric type volutes had been employed in this investigation, which is shown in Figure 2, where (a) volute 1, had been designed to neglect the effects of friction and (b) volute 2, had been designed to incorporate effects of friction.



(a) (b)

Figure 2. Asymmetric volute models (a) volute 1 designed to neglect frictional effects (b) volute 2 designed to incorporate frictional effects

Polyhedral meshing shown in Figure 3 had been employed during this study which provided the advantages of using less mesh elements, which in turn, decreased computation time as well as cost and increased the accuracy of the results.

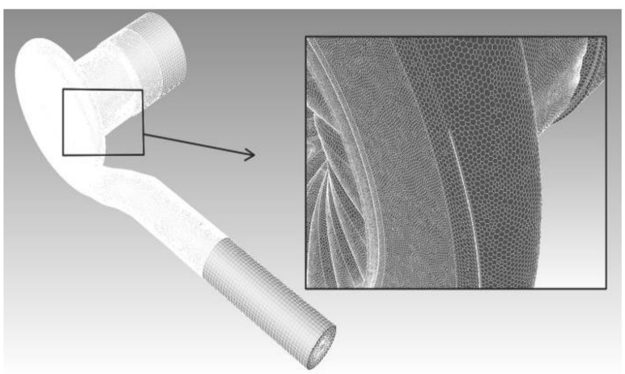
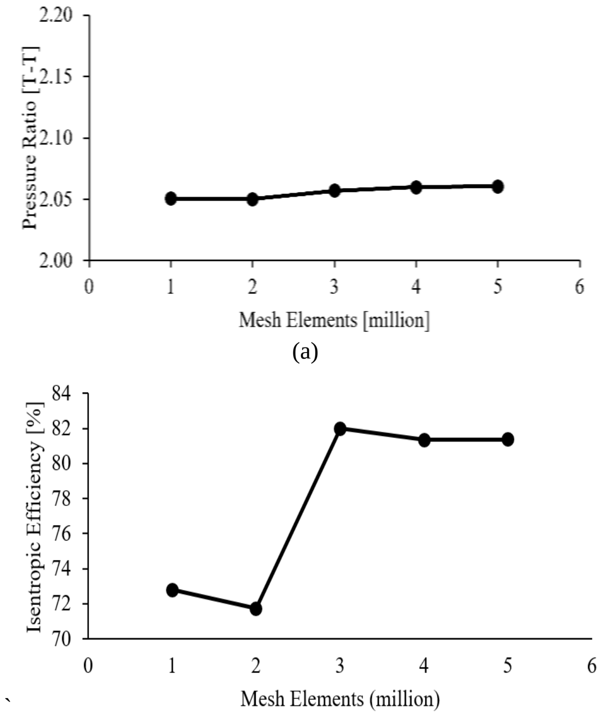


Figure 3. Polyhedral meshing of asymmetric volute models

Figure 4 depicts the results of the conducted mesh independence study, showing negligible variations after four million elements. Hence, this is the number of mesh elements that had been used throughout this investigation.

Three dimensional Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations have been utilised with continuity, momentum and energy equations to model all turbulence fields [4], hence this has been employed to numerically solve the simulations of this study. Double precision has been used with SIMPLE scheme for the pressure-velocity coupling and Green Gauss Node Based for spatial discretization gradient with second order for greater accuracy. The two-equation turbulence model Shear Stress Transport (SST) has been employed for its accuracy, robustness and superiority in flow separation between the free-stream using k-ε algorithm and near the boundary using k-ω algorithm [5, 6]. Air ideal gas has been selected as the working fluid to carry out this compressible flow investigation with a dimensionless operating impeller speed of 98.2. Boundary conditions have been specified with no-slip and wall roughness on all walls, as well as mass flow rate at the inlet and pressure at the outlet of the best efficiency point from the compressor map. In addition to this, turbulence intensity of 5% and 10% had been specified at the inlet and outlet, respectively [7, 8]. Furthermore, a time step size equivalent to 3° impeller rotation with 120 time-steps has been chosen to solve a single revolution.



(b)

Figure 4. Mesh Independence Study (a) Pressure Ratio (b) Isentropic Efficiency

3. RESULTS & DISCUSSION

Figure 5 illustrates the locations of various planes through the turbocharger volute that have been created from the first cross-section downstream of the tongue, denoted by FP to the volute outlet, denoted by OUT. In addition to this, the cut-off location is denoted by CO, beginning of the exit cone is denoted by EC and the beginning of the discharge duct is denoted by DD.

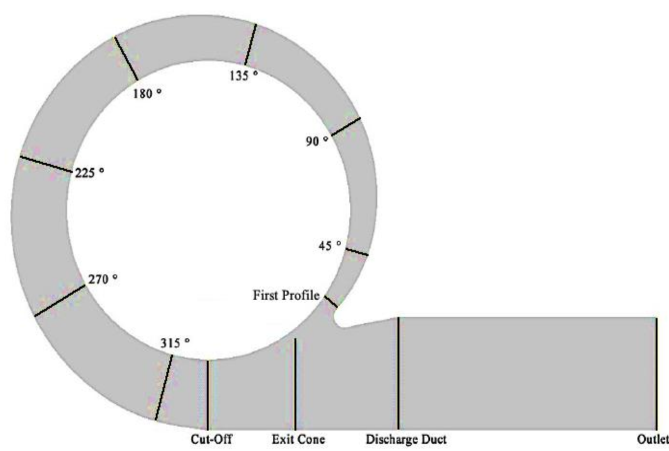
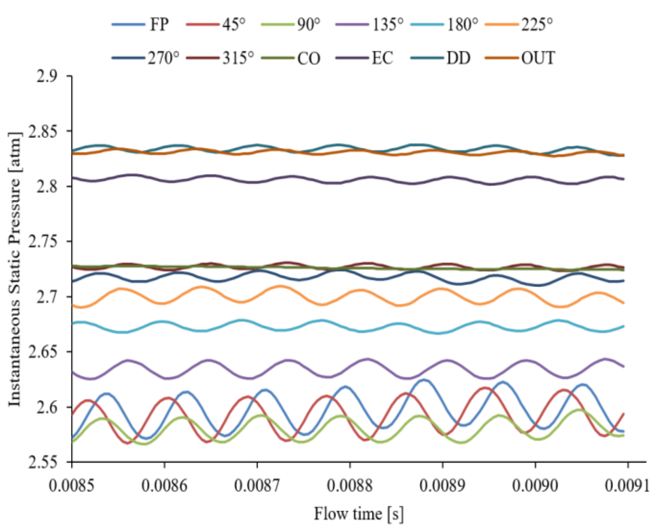


Figure 5. Cross-sectional planes created through the volute

Figure 6 depicts the instantaneous time-averaged static pressure fluctuations through cross-sections around the turbocharger volute over one revolution. Figure 6(a) displays this for volute designed to neglect frictional effects and Figure 6(b) displays this for volute designed to incorporate frictional effects. Both figures also displays their corresponding table detailing the minimum, maximum, mean and standard deviation of instantaneous static pressure across one revolution. It can be seen that there are many overlaps in instantaneous static pressure between the profiles in the case of Figure 6(a), whereas in the case of Figure 6(b), instantaneous static pressure clearly increases as the flow travels downstream of the turbocharger volute.

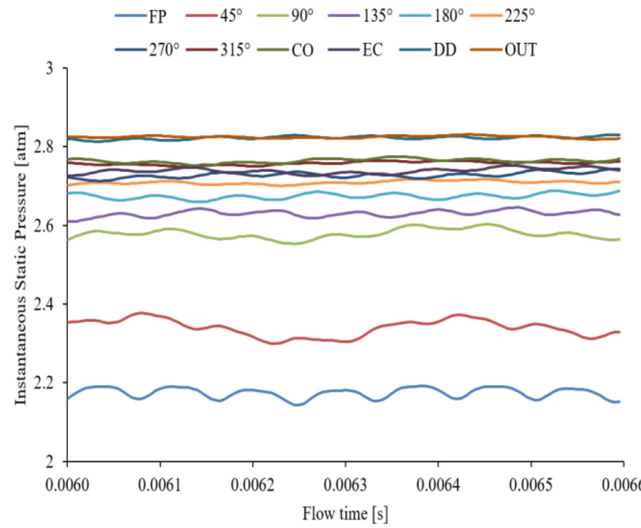
In Figure 6(a), the overlaps in instantaneous static pressure is exhibited from FP to 90°. From 135° to 270° instantaneous static pressure increases as expected prior to depicting overlaps at 315° and CO. Instantaneous static pressure increases further at EC, which then overlaps once more with DD and OUT. The instantaneous static pressure at FP reaches higher than instantaneous static pressure exhibited at 90° at its maximum. At 90°, instantaneous static pressure reaches lower than the instantaneous static pressure observed at FP at its minimum. From 135° to 270°, a prominent instantaneous static pressure increase downstream of the turbocharger volute is displayed. Instantaneous static pressure at 315° is viewed to be higher and lower at its maximum and minimum, respectively, compared to that at CO. Additionally, instantaneous static pressure at EC is higher and is seen to increase as the fluid travels downstream to OUT. Figure 6(a) table shows that minimum instantaneous static pressure is from FP to 90°, while the maximum instantaneous static pressure is at the DD and OUT. Moreover, the maximum standard deviation of instantaneous static pressure is localised at 45°.

In Figure 6(b), it can be seen that instantaneous static pressure linearly increases as the flow travels downstream the volute. Figure 6 (b) table shows that minimum instantaneous static pressure is exhibited at the FP, while the maximum instantaneous static pressure is also at the DD and OUT, similar to volute 1. Furthermore, the maximum standard deviation of instantaneous static pressure is localised at 45°, also similar to volute 1.



	FP	45°	90°	135°	180°	225°	270°	315°	CO	EC	DD	OUT
Min	2.57	2.57	2.57	2.63	2.67	2.69	2.71	2.72	2.72	2.80	2.83	2.83
Max	2.62	2.62	2.60	2.64	2.68	2.71	2.72	2.73	2.73	2.81	2.84	2.83
Mean	2.60	2.59	2.58	2.63	2.67	2.70	2.72	2.73	2.73	2.81	2.83	2.83
σ	0.02	0.01	0.01	0.01	0.00	0.01	0.00	0.00	0.00	0.00	0.00	0.00

(a)



	FP	45°	90°	135°	180°	225°	270°	315°	CO	EC	DD	OUT
Min	2.14	2.30	2.55	2.61	2.66	2.70	2.71	2.75	2.75	2.73	2.81	2.82
Max	2.19	2.38	2.60	2.65	2.69	2.72	2.74	2.76	2.78	2.75	2.83	2.83
Mean	2.17	2.34	2.58	2.63	2.67	2.71	2.73	2.76	2.76	2.74	2.82	2.82
σ	0.01	0.02	0.01	0.01	0.01	0.00	0.01	0.00	0.01	0.01	0.00	0.00

(b)

Figure 6. Instantaneous static pressure through the centrifugal compressor (a) volute 1 (b) volute 2

Table 1 details the percentage difference of the minimum, maximum and mean instantaneous static pressures between volute 1 and volute 2 through the cross-sections. It is observed that no variations in minimum instantaneous static pressures are depicted at 270°, no variations in maximum instantaneous static pressures are depicted at the volute outlet and no variations in the mean instantaneous static pressures are depicted from 90° to 180°. The highest variations in minimum, maximum and mean instantaneous static pressures are identified at FP in the favour of volute 1 with 16.7%, 16.4% and 16.5%, respectively. In addition to this, average instantaneous static pressures are in the favour of volute 1 at FP, 45°, EC, DD and OUT, whereas instantaneous static pressures are in the favour of volute 2 at 225°, 270° and CO.

Table 1. Percentage difference of instantaneous static pressure

	Percentage difference of instantaneous static pressure[%]											
	FP	45°	90°	135°	180°	225°	270°	315°	CO	EC	DD	OUT
Min	16.73	10.51	0.78	0.76	0.37	-0.37	0.00	-1.10	-1.10	2.50	0.71	0.35
Max	16.41	9.16	0.00	-0.38	-0.37	-0.37	-0.74	-1.10	-1.83	2.14	0.35	0.00
Mean	16.54	9.65	0.00	0.00	0.00	-0.37	-0.37	-1.10	-1.10	2.49	0.35	0.35

4. CONCLUSION

This presented numerical study focuses on the operational effectiveness of high-pressure centrifugal compressor volute tailored for turbocharger applications. This study has been carried out using CFD based techniques with a commercial code, FLUENT®. Two asymmetric volutes, volute 1 and volute 2, had been investigated for this study with the same compressor stage. Volute 1 had been designed by neglecting the effects of friction and volute 2 had been designed by incorporating the effects of friction. The higher the pressure through the volute, the more effective the operation of the volute is. It can be seen that instantaneous static pressure fluctuations through volute 1 is highest in the first profile, followed by that at 45° and the exit cone in comparison with volute 2. This suggests that the influence of friction plays a key role in the operational effectiveness and therefore designing a volute to neglect the frictional effects for high-pressure centrifugal compressor means the operational effectiveness is overestimated. Future work recommendations are to investigate how velocity magnitude and temperature behaves with respect to the operational effectiveness of the volute specifically designed for turbocharging applications.

NOMENCLATURE

D_{out}	Diameter at the outlet
FP	First Profile
EC	Exit Cone
DD	Discharge Duct
OUT	Volute Outlet

REFERENCES

[1] Parrondo-Gayo, J. L., Gonzalez-Perez, J., & Fernández-Francos, J. (2002). The effect of the operating point on the pressure fluctuations at the blade passage frequency in the volute of a centrifugal pump. Journal of fluids engineering, 124(3), 784-790.

[2] Ballesteros-Tajadura, R., Velarde-Suárez, S., Hurtado-Cruz, J. P., & Santolaria-Morros, C. (2006). Numerical calculation of pressure fluctuations in the volute of a centrifugal fan. Journal of Fluids Engineering, 128(2), 359-369.

[3] Kaupert, K. A., & Staubli, T. (1999). The unsteady pressure field in a high specific speed centrifugal pump impeller—part I: influence of the volute. Journal of Fluids Engineering, 121(3), 621-626.

[4] Sharma, S., Broatch, A., García-Tíscar, J., Allport, J. M., & Nickson, A. K. (2017). Acoustic characteristics of a ported shroud turbocompressor operating at design conditions. *International Journal of Engine Research*, 1468087418814635.

[5] Mojaddam, M., Benisi, A. H., & Movahhedi, M. R. (2012, June). Investigation on effect of centrifugal compressor volute cross-section shape on performance and flow field. In ASME Turbo Expo 2012: Turbine Technical Conference and Exposition (pp. 871-880). American Society of Mechanical Engineers.

[6] Sendanayake, I., Asim, T., & Mishra, R. (2017). Development of novel vehicle design modelling functions for quantifying vehicles' shapes. International Journal of Research, Science, Technology & Management, 7(2), 26-31.

[7] ANSYS, I., ANSYS FLUENT User's Manual Release 15.0, I. ANSYS, Editor. 2011: U.S.A.

[8] Samale, A. and J.E. Pacheco. Volute CFD modeling evaluation for centrifugal compressors. in ASME Turbo Expo 2014: Turbine Technical Conference and Exposition. 2014. American Society of Mechanical Engineers.