Journal of Applied and Computational Sciences in Mechanics, Vol. 28, No. 1, 2017.

Experimental and Numerical Flow Investigation through a Radial Flow Compressor Volute

M. Mojaddam[1], A. Hajilouy Benisi[2]

1. Introduction

Having a high pressure ratio, appropriate efficiency and broad operating range are the main concerns for the radial flow compressor designers who need a good understanding of flow behavior in compressor components including impeller, diffuser and volute.

As the flow in a compressor volute is turbulent and fully three dimensional, performance prediction of this component requires more attention using 3D models where the previous 1D and 2D models are not satisfactory.

Besides the development of optical measurement techniques, conventional measuring instruments such as three-hole probes for measuring pressure and velocity magnitude and also five-hole probes for measuring the pressure and velocity vector are widely used since their results are precise and acceptable.

In the current study, the flow field is investigated experimentally and numerically through a semi external volute of a radial flow compressor in a turbocharger.

After obtaining the compressor component geometries, creating the model and meshing the parts, flow field simulation is performed by solving the Reynolds averaged Navier-Stokes (RANS) equations using the SST turbulence model.

Volute performance is obtained using pressure taps on compressor spiral case to evaluate the volute losses at different operating conditions. Furthermore, the flow field is explored by implementing a five-hole probe traversing at different cross sections of the volute.

2. Modeling

For compressor modeling, all components including inlet, impeller, vane-less diffuser and volute are considered. Multi-Slice CT scan of the spiral case and impeller are performed to obtain the geometries. Constructing the geometry of the impeller needs more attention. Hence the impeller is re-designed to meet the geometry requirements which are obtained through scanning analysis.

Creating the compressor component models, the mapped structured hexahedral elements are implemented for the impeller and vane-less diffuser section grids and the unstructured tetrahedral elements for volute meshes (Fig. 1).

For performance prediction simulation, coarse grids are adequate since they can lead to decreased cost of numerical calculations. However, finer meshes are required for flow field investigations. Hence, both coarse and fine grids are explored to satisfy the mesh independency requirements.

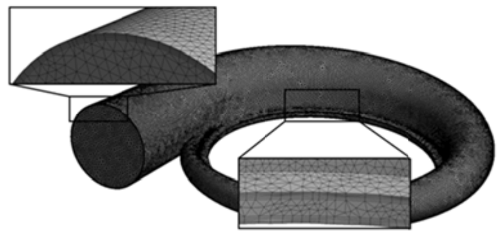

Fig. 1. Volute mesh

A three dimensional viscous flow solver is used in which the Reynolds averaged Navier-stokes equations are solved using the shear-stress transport (SST) turbulence model.

The discretization of the equations is done via segregated implicit method and the SIMPLEC algorithm is used for coupling pressure and velocity.

3. Experimental study

The test rig at the Sharif University of Technology's turbocharger laboratory is used to perform the experimental studies.

Total and static pressures are measured by three precision strain gauge transducers and 'scanivalve' channel selector system. Calibrated J type thermocouples are used to measure temperatures.

The appropriate stations are considered for measurement devices to obtain the desired data at each compressor components.

4. Result and discussion

4.1. The effect of volute on pressure losses

Pressures and temperatures are measured at the inlet and discharge of the compressor components including the impeller, diffuser and volute, in order to study the effect of each component in total pressure losses.

The sample results are presented here in Table 1. The numerical and experimental results are the total pressure losses at the volute for 0.05, 0.08 and 0.11 kg/s mass flow rates at the rotational speed of 60,000.

It is clearly seen that the deviations of numerical and experimental results are decreased with increasing of the rotational speed.

Table 1. Experimental and numerical results-Total pressure loss at the volute

Mass Flow rate (kg/s)	Numerical Results (Pa)	Experimental Results (±%1.5 Error) (Pa)	Deviation (%)
0.05	2119	2300	7.86
0.08	3781	3900	3.05
0.11	5283	5500	3.94

---

[1] Corresponding Author, Assistant Professor, Mechanical Engineering and Energy Department, Shahid Beheshti University, m_mojaddam@sbu.ac.ir

[2]Professor, Mechanical Engineering Department, Sharif University of Technology

4.2. Flow field Study

For measuring the pressure and velocity through the volute cross section, four stations are considered at the volute casing. Traversing a five-hole probe at each station, the flow field characteristics are obtained. Fig. 2 shows the installed probe at one of the measurement stations with its traversing mechanism.

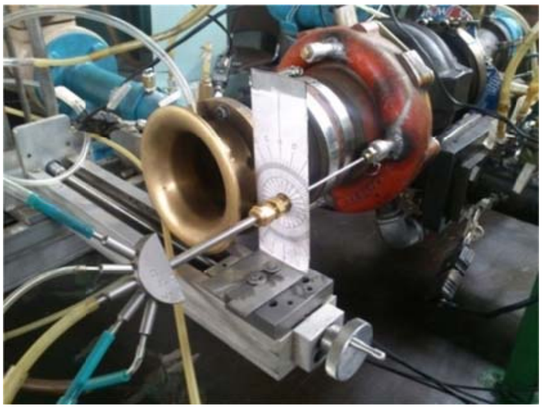

Fig. 2. The five-hole probe and traverse mechanism installed at the compressor test rig

Using the five-hole probe, five pressure values are measured which are then used for obtaining the total pressure, static pressure, velocity magnitude and the yaw and pitch angles to be used in the velocity vector calculation.

The typical results are shown in Fig. 3 and Fig. 4. The numerical and experimental values of velocity ratios (the magnitude of velocity to the mean of velocity at the cross section) are plotted at different traverse positions. The results are reported for two different conditions with 0.15kg/s and 0.13 kg/s mass flow rates at 60,000 rpm and 70,000 rpm, respectively.

The results show a good agreement between the numerical results and the experimental ones. The velocity vector profiles are used for assessing the fluid structure inside the volute and are further used to recognize the loss mechanism in the volute.

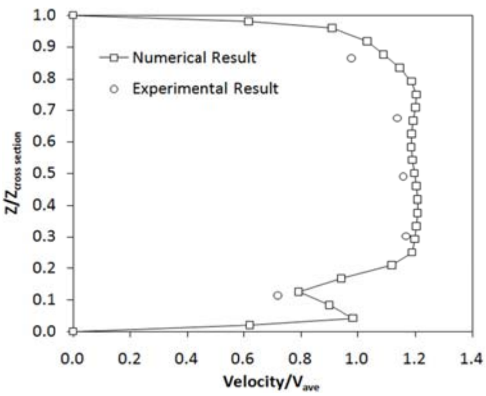

Fig. 3. Numerical and experimental results- the velocity profile at 0.15 kg/s mass flow rate

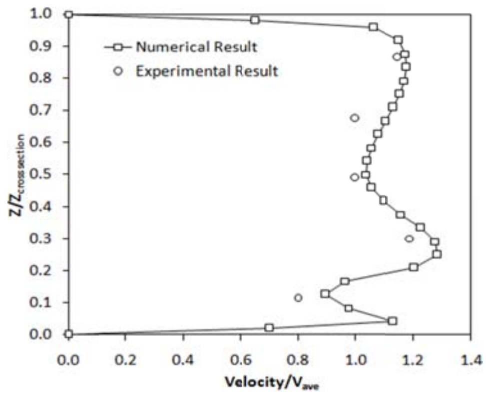

Fig. 3. Numerical and experimental results- the velocity profile at 0.13 kg/s mass flow rate

5. Conclusion

In this research, experimental and numerical flow investigations are performed through a radial flow compressor volute in order to recognize the flow structure and its effect on volute pressure losses.

Flow field investigation through the volute cross sections at different operating range of the compressor is performed utilizing a five-hole probe.

Numerical studies are verified by measurement data which shows good consistency.

The results show that the flow pattern at each volute cross section is a forced vortex structure generated from the radial component of the velocity vector at the volute inlet. However, the diffuser's discharge flow changes the structure at the hub wall. The effect of diffuser discharge on the fluid structure in semi-external volute cross section is recognized as an extra pressure loss source in this type of volute.