International Journal of Engineering Research and Development
e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 10, Issue 11 (November 2014), PP.77-83

An Investigation on the Performance Characteristics of a Centrifugal Compressor

L. G. Das[1], P. K. Sen[2]
[1]*PPE, CSIR- Central Mechanical Engineering Research Institute, M.G. Avenue, Durgapur -713209*
[2]*PPE, CSIR- Central Mechanical Engineering Research Institute, M.G. Avenue, Durgapur -713209*

Abstract:-The design and off-design performance characteristics of single stage centrifugal compressor consisting of 12 vanes impeller interfacing with 11 vanes diffuser have been studied experimentally and numerically. The impeller has been designed and developed with radial exit, 30^o inlet blade angle (with tangent), 77 mm diameter and the discharge volute considering constant mean flow velocity. The performance of the compressor at varying capacity (60 to 120 % of design) by controlling the discharge valve and with the variation of rotating speed (15000 to 35000 rpm) by regulating speed of the coupled gas turbine has been conducted at the recently developed test rig. The numerical simulation has been done by adopting viscous Reynolds Average Navier-Stokes (RANS) equations with and without Coriolis Force & Centrifugal Force in rotating reference frame (impeller) and stationary reference frame (casing) respectively utilizing CFD software Fluent 14. The flow around a single vane of impeller interfacing with single vane of diffuser, the rotational periodicity and sliding mesh at the interfacing zone between rotating impeller and stationery diffuser are considered. Non dimensional performance curves derived from experimental and numerical results are presented and compared. The numerical results are found to match very closely with the experimented data near the design point and deviation is observed at the both side of the designed operating point. Non-uniform pressure profiles towards the impeller exit and strong cross flow from blade to blade are detected at low flow operating conditions. Total pressure, static pressure and velocity distributions at design and off design operation obtained from the CFD results are analysed and presented here.

Keywords:- Impeller, Diffuser, Volute, CFD, Flow

I. INTRODUCTION

Centrifugal compressors are mostly utilized in chemical industries and power plants etc., to raise the pressure of the air or chemical gas to meet the system requirements. The flow through the rotating impeller and stationery diffuser and especially at the interface region are usually affected by secondary flow phenomenon like boundary layer separation from blade suction surface, cross flow from pressure to suction surface, end wall vortices and slip of the impeller exit whirl velocity. The secondary flow phenomena are observed to affect the performance of the compressor mostly at off design operating conditions. Many researchers attempted to analyse the internal flow phenomenon of the centrifugal compressor and to predict the performance characteristics. R. Numakura[1] observed from numerical analysis that two stage centrifugal compressor for small turbocharger has increased operating range. Y. Wang and Z. Luo[2] noticed from their CFD results that operating mass flow range of the compressor increases with increase of size of the compressor and same decreases with increase of operating speed. Arunachalam et al. [3] studied the effect of impeller trailing edge skew angle on stall and pressure ratio, they concluded that proper blade skew angle of the impeller is very important for higher pressure ratio or higher stall margin. Ramamurthy et al. [4] analysed the optimum the diffuser setting angle for maximum stage efficiency of centrifugal compressor.

In this paper, performance characteristics of a compressor driven by an inward radial flow gas turbine have been presented. 3-D flow domain around a single vane of impeller, single interfacing vane of diffuser and 2-D dimensional flow domain of volute have been modelled [5] out in Gambit2.4.6. The internal flow phenomenon has been analysed with CFD software [6], Fluent 14.

II. DETAILS OF THE COMPRESSOR

The centrifugal compressor is coupled with inward radial flow gas turbine and it breaks the gas turbine from no load operation. The impeller of the compressor is radial bladed and interfaced with diffuser and enclosed in volute. The common shaft is supported by aerostatic journal bearing and pad thrust bearings

Table -1 Compressor Details	
Fluid	**Air**
Capacity	**0. 1 kg/s**
Pressure ratio	**1.6**
Max Operating speed	**40000 rpm**
Impeller inlet diameter	**47 mm**
Impeller outlet diameter	**77 mm**
Impeller inlet vane angle	**30 ^{o}**
Impeller outlet diameter	**90 ^{o}**
Suction diameter	**38 mm**
Discharge diameter	**25 m**

III. EXPERIMENTATION

An experimental setup which consist a turbine coupled with compressor is developed to conduct the experiments, as shown in Fig. 1. The compressor sucks atmospheric air through 80 mm (NB) pipe line and discharges to 25 mm (NB) pipe line to the atmosphere. The flow of air in discharge pipe line is controlled by a ball valve and also a 12 mm (NB) bypass line is connected to discharge line for avoiding surge. The suction line consists an orifice meter for measurement of air inflow with a differential pressure transmitter, a pressure transmitter for inflow pressure measurement and a temperature transmitter for inflow temperature measurement. The discharge line is equipped with a temperature transmitter and a pressure transmitter for discharge temperature and pressure measurement respectively. The operating speeds are measure by infrared speed sensor. The turbine was operated at cold condition by compressed air of 2 bar, 1.75 bar and 1.5 bar inlet pressure and at the speed of 35000, 20000 and 15000 rpm.

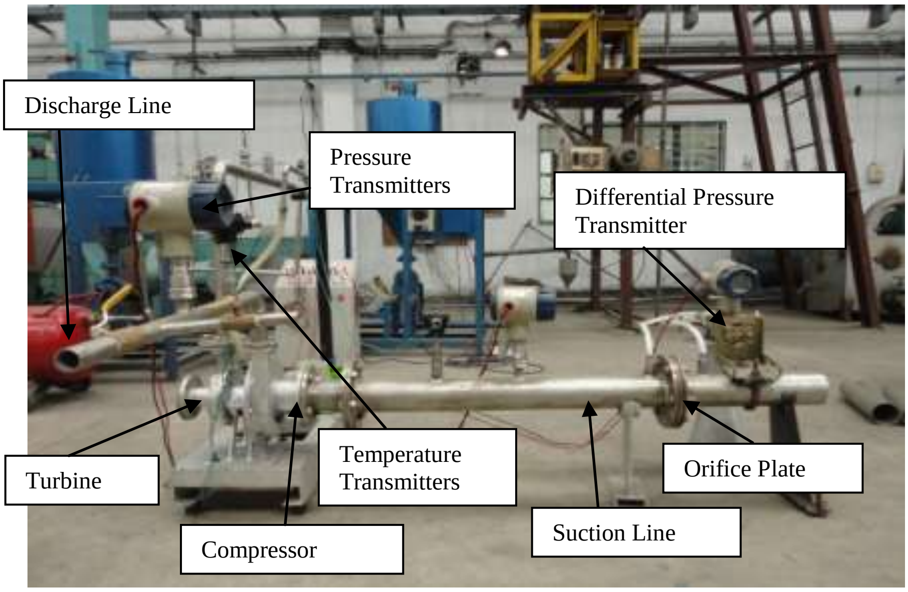

Fig. 1: Experimental Rig

3.1 Experimental Uncertainty

One calibrated differential pressure transmitter make Honeywell Model ST 3000, Range 0 to 10000 mm of water with accuracy ± 0.0375 % is used for measurement of suction air differential pressure across the orifice plate . For air pressure measurement in suction and discharge lines two numbers of pressure transmitters make Honeywell Model ST 3000, Range 0 to 35 bar and accuracy ± 0.0375 % are used. Two numbers of calibrated temperature transmitters make Honeywell, Model STT 25M, range -200^{o}C to + 450^{o}C and resolution ± 0.05 % are used for measurement of air temperature in suction and discharge line respectively. The rotational speeds of the compressor shaft are measured by infra red speed sensor; ACT-3, Monarch Instrument, range 5 to 99999 rpm, accuracy 0.0015% and all the data are stored in a computer

IV. NUMERICAL ANALYSIS

4.1 Grid Generation

The 3-D flow domains around a single vane of impeller and diffuser have been discretised with unstructured tetrahedral grid (Fig. 2). Grid size of the flow domain is varied from 0.3 mm near the vane surface to 3 mm in the core flow region. Size function has been utilised to create fine grid near vane surface. There are 242442 tetrahedral cells in the impeller and 151512 tetrahedral cells in the diffuser flow domain. Grid independence has been checked with 0.1 mm grid near wall surface to 1 mm grid at the core flow and numerical results starts converging and overlapping from 3.5×10^5 grids. The 2-D fluid flow domain of the volute has been discretised with grid size 0.3 mm near the vane surface to 3 mm in the core flow region. There are 14436 quadrilateral cells in the in the 2-D flow domain of the volute.

4.2 Numerical Scheme

The CFD analysis of the compressor has been carried out in [6] Fluent 14. The 3-D fluid flow (air) through the impeller and diffuser and the interaction between rotating impeller and stationary diffuser has been analysed. The computational flow domains of impeller and diffuser are rotationally periodic in nature. The fluid flow around a single vane of impeller interfacing with a diffuser vane has been solved. The interfacing zone i.e. between the impeller outlet and diffuser inlet has been considered as sliding mesh for unsteady analysis. The numerical simulations are done by adopting viscous unsteady Reynolds Average Navier-Stokes (RANS) equations with and without Coriolis Force & Centrifugal Force in rotating impeller and stationery casing respectively. The energy equations are also solved for considering the variation of air density.

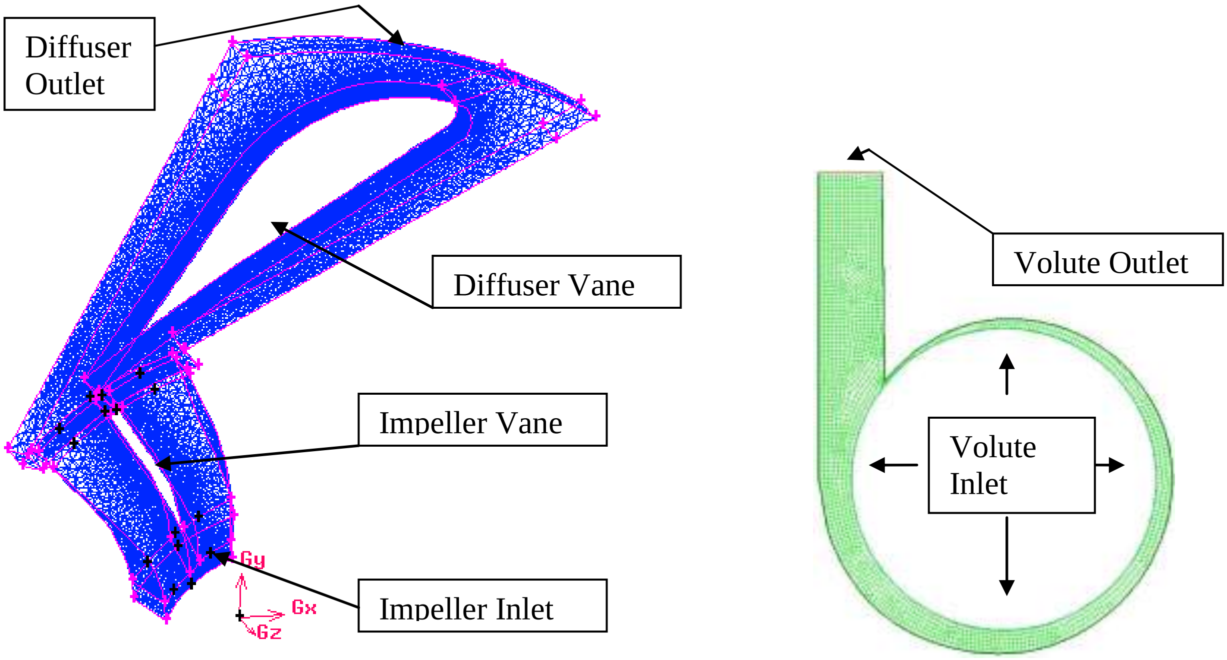

Fig . 2: 3-D Grid for impeller and diffuser, 2-D grid for volute

4.3Boundary Condition

The boundary condition at the impeller inlet has been specified as pressure inlet and diffuser outlet as pressure outlet. The inlet pressure has been considered as zero gauge pressure and diffuser outlet pressure is varied for nearest to the expected pressure at the operating speed. No slip condition for impeller blades and walls with respect to the impeller rotating flow domain are specified. No slip conditions are also specified for diffuser wall and blades with respect to stationary flow domain. The mass flow rate at impeller inlet and diffuser outlet has been checked and converged for the desired solution. The fluid flow through the volute has been studied separately and the numerical results obtained from impeller and diffuser are utilised for the volute simulation as boundary condition. 2-D flow has been solved for the volute as there is no geometrical variation in the axial direction of the volute from inlet to discharge.

V. TEST RESULTS AND DISCUSSION

5.1 Performance Parameters

The compressor flow passage simulations are carried out with number of points from minimum flow to maximum flow with variation of discharge pressure at diffuser. The compressor mass flow rate obtained for considered single flow domain is multiplied by number of passage in the impeller and diffuser. The mass weighted average total discharge pressure at the outlet area of the discharge volute and the mass weighted total suction pressure at impeller inlet is obtained from the Fluent simulation result. The pressure ratio is defined as.

$$r = \frac{P_{03}}{P_{01}} \tag{1}$$

The mass flow rate is non dimensionalised as explained below

$$\overline{m} = \frac{\dot{m}\sqrt{T_{01}}}{P_{01}} \tag{2}$$

The compressor performance curve has been drawn with pressure ratio against non dimensional mass flow rate for various operating speed. The predicted and the experimental performance curve are presented in Fig.-3. The performance curve has been plotted with total pressure ratio against non-dimensional mass flow parameter for three different speeds.

The predicted non dimesionalised flow and pressure ratio curve is closely matching near maximum pressure ratio and at stalled condition. with experimental results and significant surging characteristics has been found as the flow reduces from nominal flow rate to minimum flow rate .

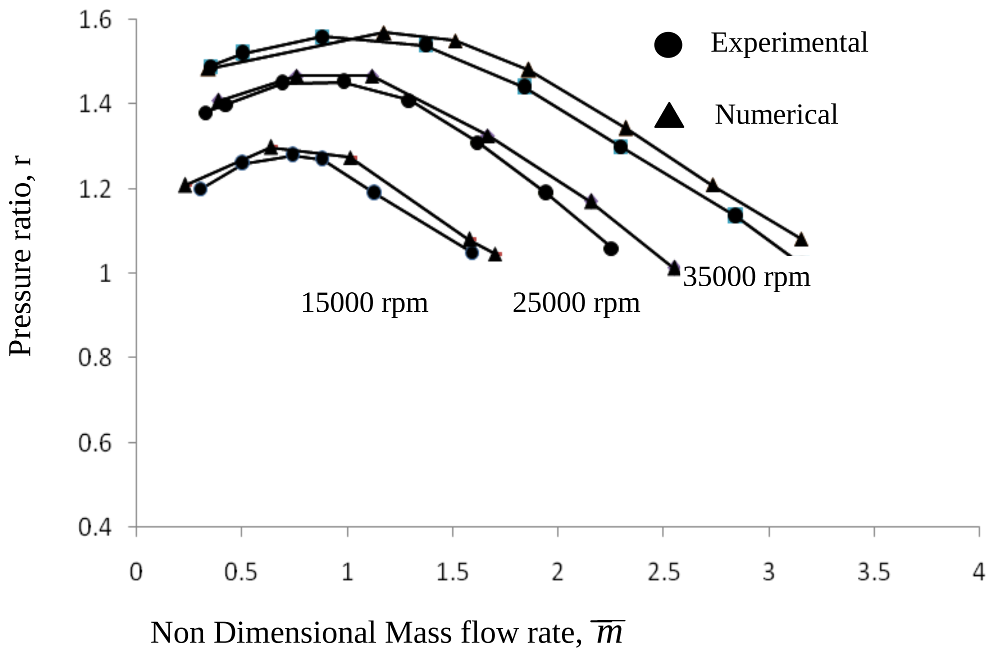

Fig. 3: Performance Curve

The numerical performance curve is slightly over predicted than the experimental curve at high flow operating conditions and the reason may be the understimation of some energy losses in the numerical analysis which arrises due to secondary flow phenomenon at high fluid flow rate through the compressor. The numerical total pressure distribution, static pressure distribution and velocity field of impeller and difffuser at 35000 rpm at nominal point is presented in Fig. 4 to Fig 6. A tremendous recirculation is set up at the tip of exit region of blade pressure surface and it enhances towards the low flow operation point. It is detected both in Fig.4 and Fig.5. The enegy losses due to these recirculation casues the drooping performance characteristics at low flow region.

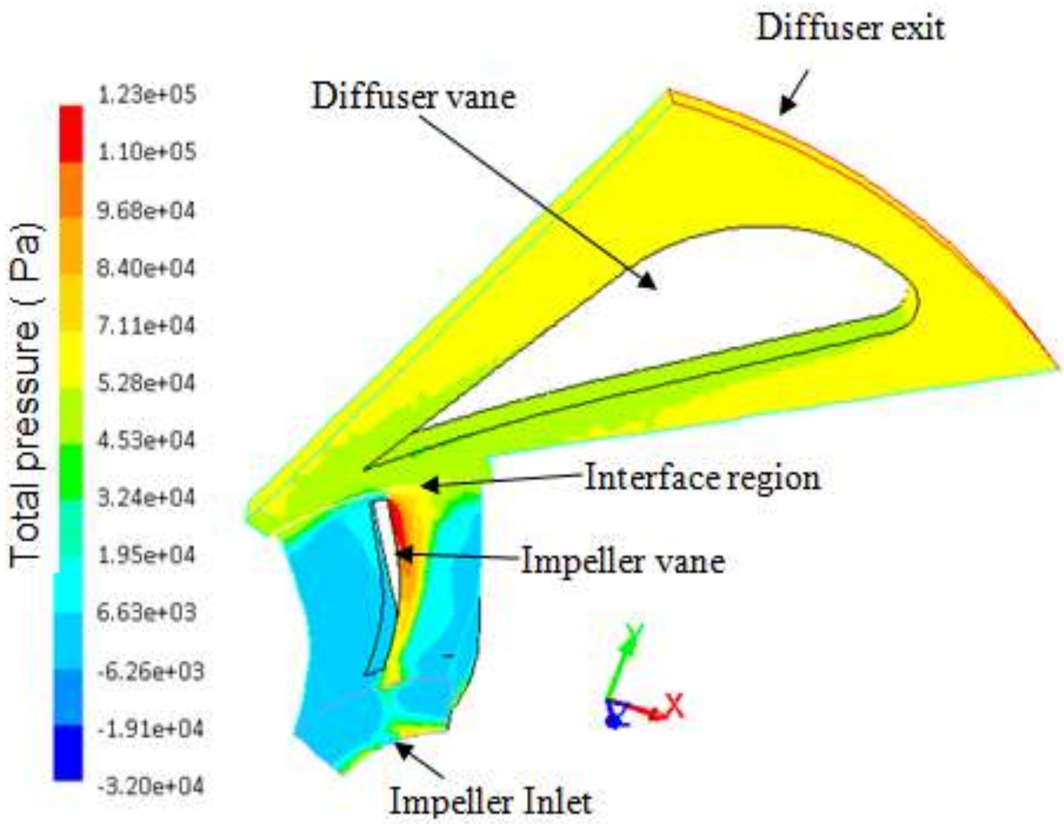

Fig. 4: Total pressure distribution

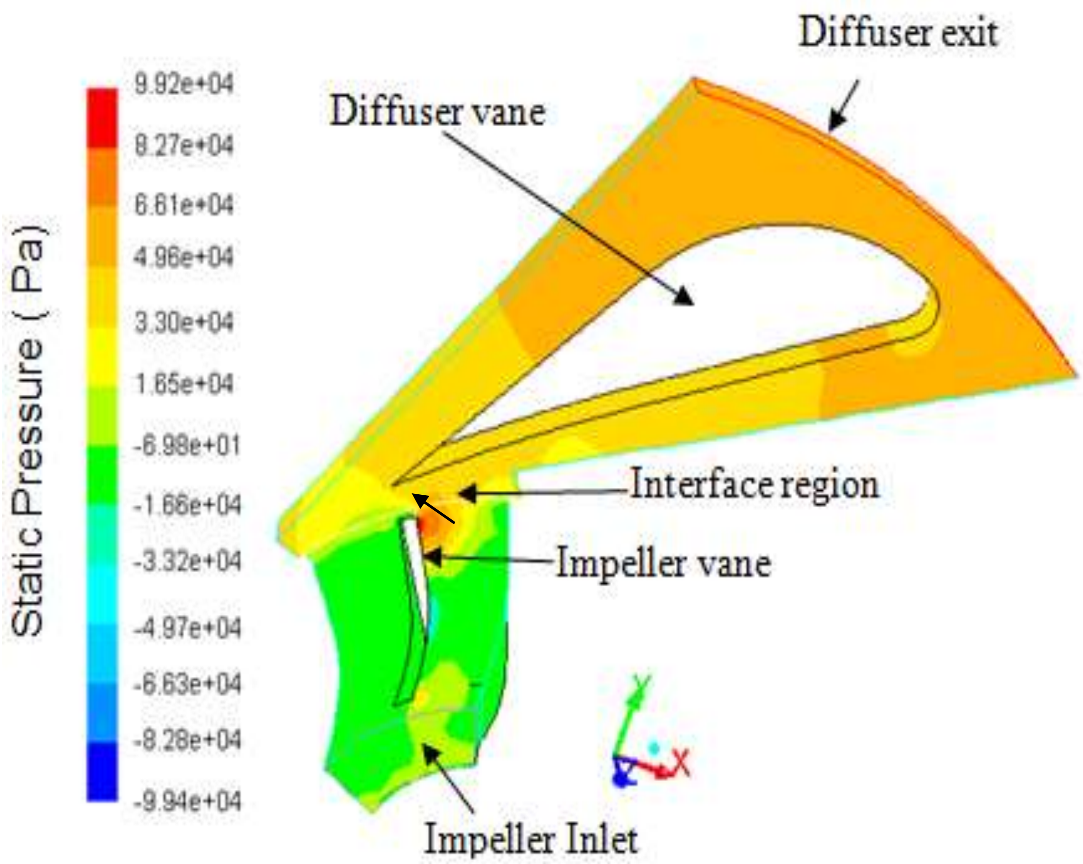

Fig. 5: Static pressure distribution

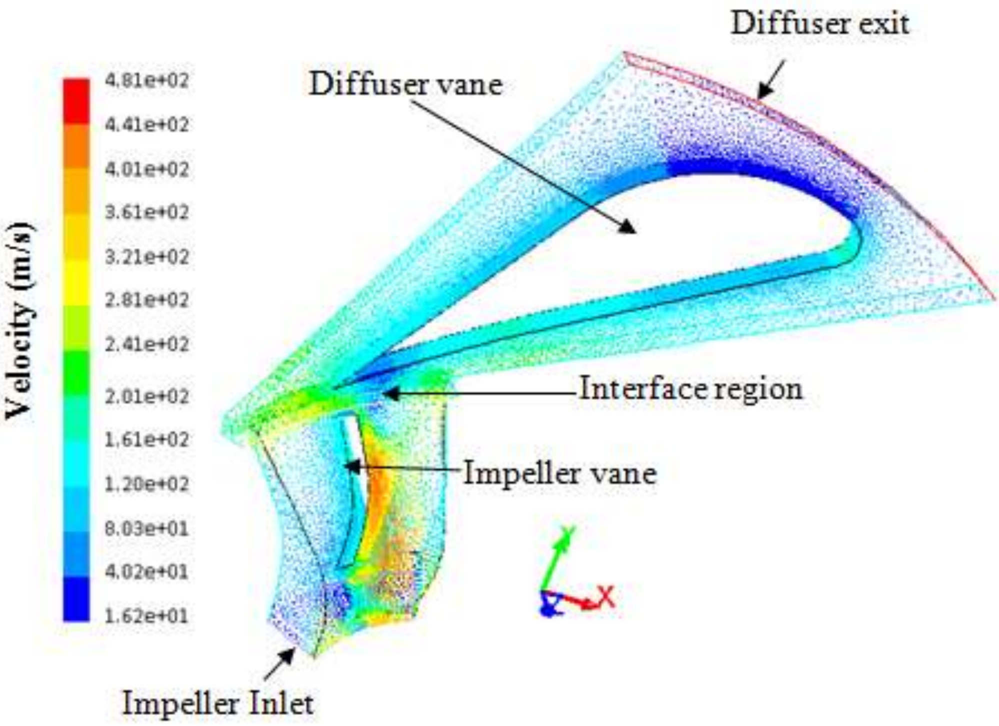

Fig. 6: Velocity Vectors

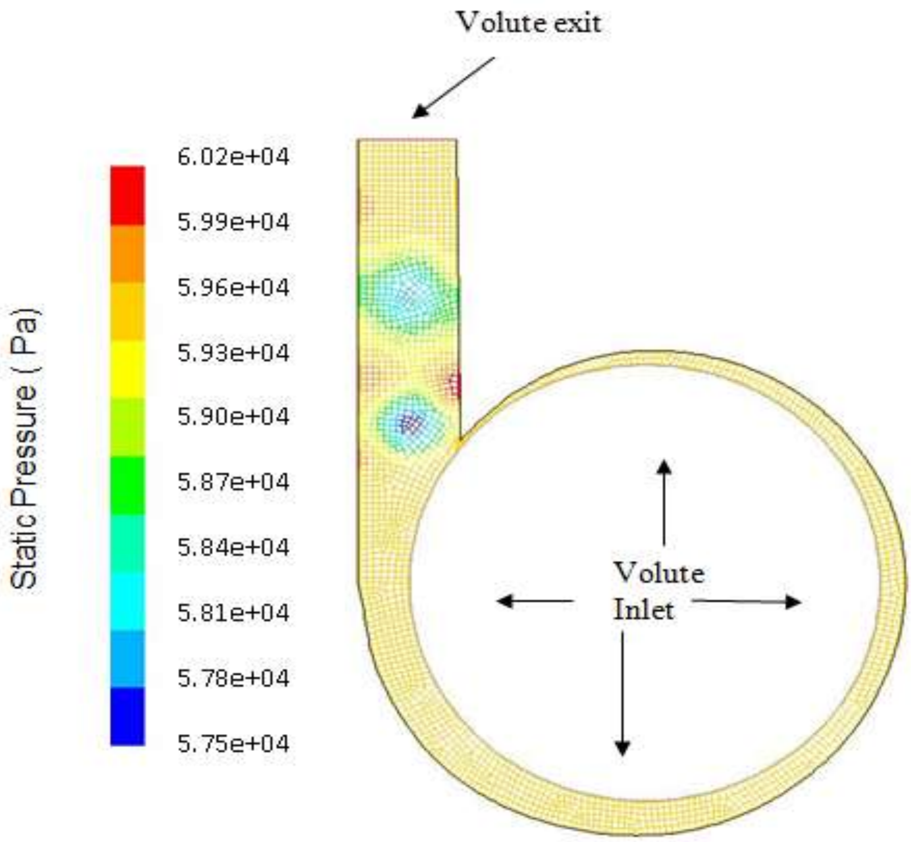

Fig. 7: Static pressure distribution of volute

The fluid flow against the adverse pressure gradient causes thickening of the boundary layer at the blade suction surface leading flow separation from the wall surfaces and it eveidently occurs at low flow rate . The pressure developed by the compressor is drastically reduced at these low flow region which indicates surging of the compressor.

A cross flow i.e. from impeller vane pressure surface to suction surface is detected in numerical analysis and it increases towards the high flow operating condition from nominal operting condition. The enrgy losses as found in experimentation towards high flow operation condition is higher than the predicted values the reasson may be underestimation of energy losses caused due to these secondary flow phenomenon.

The 2-dimensional flow analysis of the volute reveals that static pressure distribution throughout the volute is almost uniform. It is matching with the design principle of constant mean flow velocity for the volute. Volute has been design to accommodate the all-round exit flow of the diffuser rather than static pressure recovery. A recirculation is noticed which originates after the volute tongue region and which is detected in Static pressure distribution (Fig. 7). This recirculation induces some energy losses, the variation in total pressure at the diffuser outlet and volute out let is very minimum and this difference in the order of 2000 Pa.

V. CONCLUSION

The predicted nondimension mass flow rate –pressure ratio curves are presented for the entire mass flow rate from surging line to maximum flow rate. The predicted performance curve have been compared with the experimental performance curves. This work reveals that performance characteristics of the centrifugal compressor for the entire flow range. The predicted perfomance curves at maximum pressure ratio and stalled condition reasonable match with experimental performance curve. The numerical perofmace curve is over predicted than experimental curve at the maximum flow condition.

The adopted nemerical methodolgy gives satisfactory performance prediction at the zone of higher pressure ratio. The 3-D flow phenomenon through the compressor especially throug the rotating impeller is highly complicated and and the performance of the centrifugal compressor depends on these complex internal flow. Further research is indeed needed.

NOMENCLATURE

P pressure, Pa
T Temperature, K
r pressure ratio
$\dot{m}$ mass flow rate, kg/s
$\overline{m}$ non dimensional mass flow rate
NB nominal bore

SUFFIX

01 total (stagnation) at impeller inlet
02 total (stagnation) at impeller outlet
03 total (stagnation) at volute outlet

ACKNOWLEDGEMENT

The work has been conduced in a project funded by CSIR- Central Mechanical Engineering Research Institute and authors are duly acknowledged to the Director, CSIR-CMERI, Durgapur, India for his kind support and permitting to publish this paper.

REFERENCE

[1]. R. Numakura, Design of a Two-stage Centrifugal Compressor for Small-size Turbomachinery , The 11 th Asian International Conference on Fluid Machinery and 3 rd Fluid Power Technology Exhibition, IIT Madras, Chennai, India, 2011
[2]. Y. Wang and Z. Luo, Simulation and Performance Analysis on Centrifugal Compressors of Different Dimensions and Variable Operation Speed, http://www.ieee.org, 2011
[3]. V.C. Arunachalam, Q.H. Nagpurwala M.D. Deshpande and S.R. Shankapal, Numerical Studies on the Effect of Impeller Blade Skew on Centrifugal Compressor Performance SASTech - Technical Journal , 2008
[4]. S.Ramamurthy, R.Rajendran and R. S. Dileep Kumar, Theoretical Evaluation of Flow Through Centrifugal Compressor Stage, Proceedings of the International Conference on Aerospace Science and Technology, Bangalore, India, 2008
[5]. GAMBIT User´s Manual
[6]. Fluent 14 and Fluent User’s Guide, Fluent Inc