12th International Conference on Recent Innovations in Science, Engineering and Management
Sri Venkateswara College of Engineering & Technology,Etcherla, Srikakulam (ICRISEM'18)
17th February 2018 www.conferenceworld.in ISBN: 978-93-87793-01-9

CFD ANALYSIS OF CENTRIFUGAL IMPELLER AT DIFFERENT SPPEDS AND MASS FLOW RATES

S.MANIKANTA[1], G.ROJA[2], H.VASU DEVA RAO [3]

[1]*M.Tech student, mechanical department, SISTAM engineering college, Andhra Pradesh, (India)*

[2]*M.Tech student, mechanical department,SISTAM engineering college, Andhra Pradesh, (India)*

[3]*Assitant Professor, mechanical department, SISTAM engineering college, Andhra Pradesh, (India)*

ABSTRACT

Computational analysis of a low speed centrifugal compressor is the point of interest of this project. In the thesis, focus is made on the probing of superior turbulence model for simulating three dimensional flows in centrifugal compressor. The strong channeled curvature's and intensive rotations prevalent in centrifugal compressor resulting high swirling and secondary flow nictitates choosing appropriate turbulence model for accurate performance predictions. The various turbulence models offered in FLUENT viz Spalart - Allmaras (curvature correction), Transition SST (curvature correction), Scaled Adaptive Simulations (Curvature correction with compressibility effect), Reynolds stress model (compressibility effect) were investigated presently for Eckardt Impeller. Reynolds stress model though involves higher computational time was found to be the superior model. It is essential to investigate the onset of surge and choke for completely understanding the performance of a centrifugal compressor. Choking phenomena was observed when the speed reached 16000 rpm with relative Mach number reaching unity in the impeller region. The maximum flow rate at 16000 rpm was 0.4 kg/s per blade and remained constant then 16500 rpm. Surging was founded to initiate when the back pressure has to reach 1.8 bar resulting in zero discharge.

I.INTRODUCTION

Centrifugal compressors are used in small gas turbines and are the driven units in most gas turbine compressors trains. They are integral part of petro chemical industry, finding extensive use because of their smooth operation, larger tolerance of process fluctuations and their highly reliability compared to other types of centrifugal compressors. Centrifugal compressors are used for high pressure ratios and lower flow rates compared to lower pressure ratios and higher flow rates in axial compressors. The most efficient region for the centrifugal compressor operation is in a specific speed range between 60< Ns<1500. Specific speed of more than 3000 usually require axial flow compressor.

The aim of this research is to evaluate the suitable turbulence model which asses the close behavior of the internal flows obtaining in the present geometry since as it is believed from the previous research a turbulence model plays a major role in disparity between various approaches (i.e. Experimental, Analytical and Numerical). This will be accomplished by developing a better understanding of their internal aerodynamics, specifically in regions within the engine where the airflow is strongly turned (high curvature). Curvature exists in many regions of an engine, however a specific focus will be directed towards the centrifugal compressor stage used in small scale compact engines. A further understanding of the flow physics and the effects of choke and surge in this region will lead to more compact designs, resulting in a higher efficiency and reduced weight; all to provide greener engine operation.

In addition to the probing of various turbulence models within a centrifugal stage in engine, the effects of choke and surge will also be investigated in a general sense. Curvature is present in many turbulent engineering flows, with one specific example being in turbo machinery applications. Whether it is, for example, the curvature of the blades in an axial machine, or the axial to radial transition in a centrifugal machine, curvature will exist somewhere in the system in most applications. An important part of understanding the flow physics in these machines is identifying and knowing how to deal with any strong channelled curvature effects. This becomes particularly relevant when using computational fluid dynamics (CFD) modelling techniques. Curvature introduces an extra level of complexity that can greatly affect the flow structure and turbulence 3 quantities, which needs to be accounted for when considering the implementation of turbulence models.

Turbulence modelling techniques have been studied by numerous researchers since their initial development. There are a variety of different turbulence models that are used widely for industrial applications. The main focus of this work is on a curvature corrected and compressibility effect versions of Spalart Allmaras (CC), Transition SST (CC), Reynolds stress model (CE), Scaled Adaptive Simulations (CC+CE).

II.SCOPE AND OBJECTIVES OF THE PRESENT WORK

The overall scope of this work entails comparison of various turbulence modelling with the objectives of developing an improved understanding of the flow phenomena associated with strong channelled curvature in general, determining how curvature corrected turbulence models account for these phenomena and thoroughly investigating the use of the Spalart Allmaras (CC), Transition SST (CC), Reynolds stress model (CE), Scaled Adaptive Simulations (CC+CE) models in low speed centrifugal compressor cases with high curvature. Various authors have previously investigated two of these model performance in various simplified test cases by comparing against experimental data and focus on the primarily mean quantities. This work expands on compares and come out with suitable turbulence model discussed above and further investigating the underlying flow phenomena of choke and surge with that turbulence model.

Therefore, the objectives of this work are accomplished by evaluating the best turbulence model on performance of low speed centrifugal impeller geometry. The present work contains the numerical simulation of the centrifugal impeller and the tracking of the choke and surge phenomena of the compressor.

The completion of this work is beneficial in terms of developing a deeper understanding of the underlying mechanisms in the impeller model in regards to curvature correction and compressibility effects of turbulence models specified. Moreover, it provides valuable performance characteristics in compressor design and further validates the use of the RSM turbulence model in other turbo machinery applications.

Table 1geometric parameters of impeller

Parameter	Unit	Specification
Blade numbers	-	20
Inlet radius at tip (rs)	m	0.14
Inlet radius at hub (rh)	m	0.045
Outlet radius (ro)	m	0.2
Impeller exit width	m	0.026
Blade inlet angle (β1s)	deg	60
Blade back sweep (β2)	deg	0

III.GEOMETRIC MODEL

The impeller geometry was designed in INVENTOR software where it is sliced to 18^0sector in order to reduce the number of elements which causes less computational time to converge the solution as shown in figure 1. This 18^0 sector is made into complete geometry by mentioning the number of blades to twenty with the option of cyclic symmetry present in the INVENTOR software to generate the full visual 3D model is shown in figure 2.

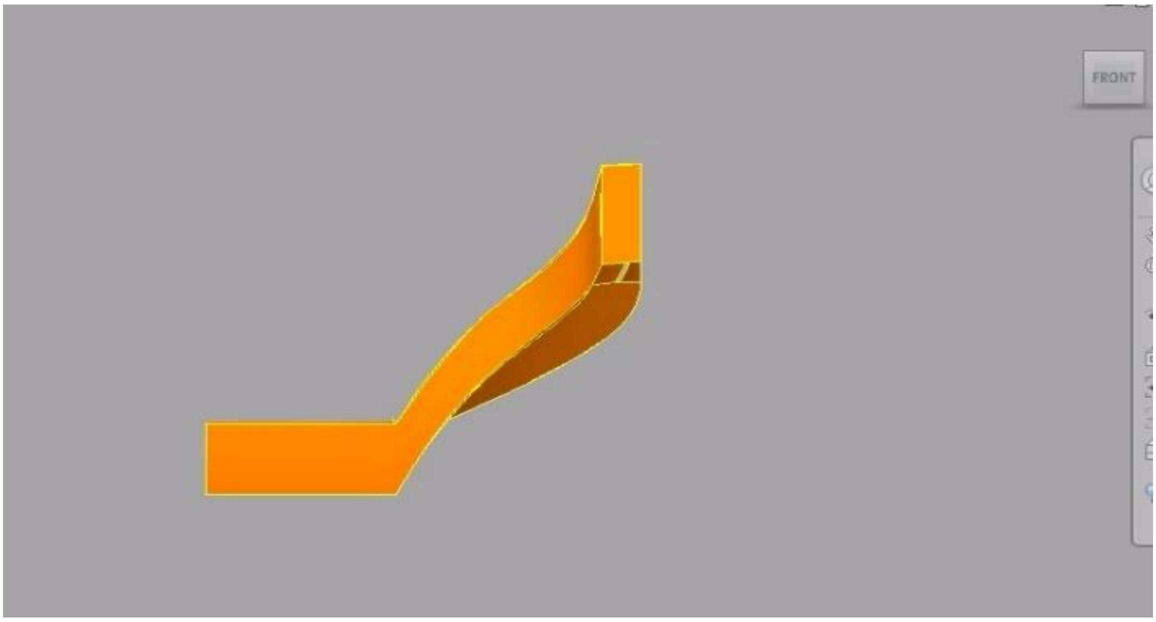


Figure 1. 18^0sector extract from compressor blade through (inventor) software.

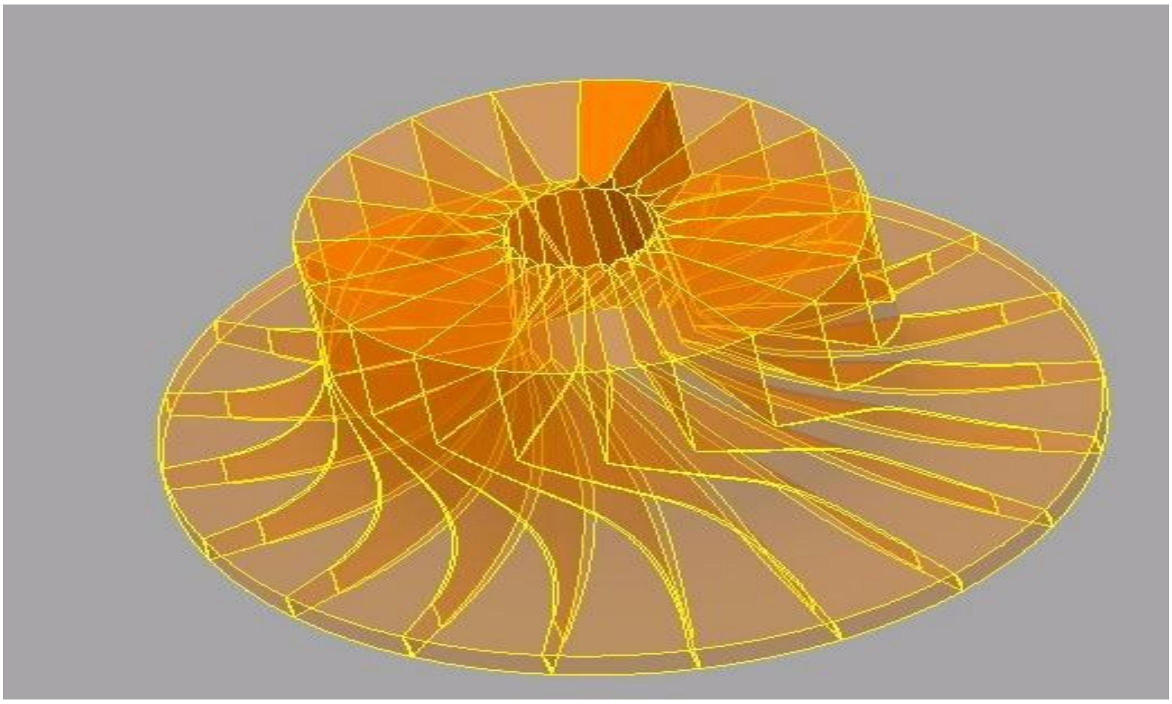


Figure 2.Compressor with 20 Blades.

3.1MESH MODEL

In the present case, the mesh obtained is structural with 28000 hexahedral elements with 31311 nodes

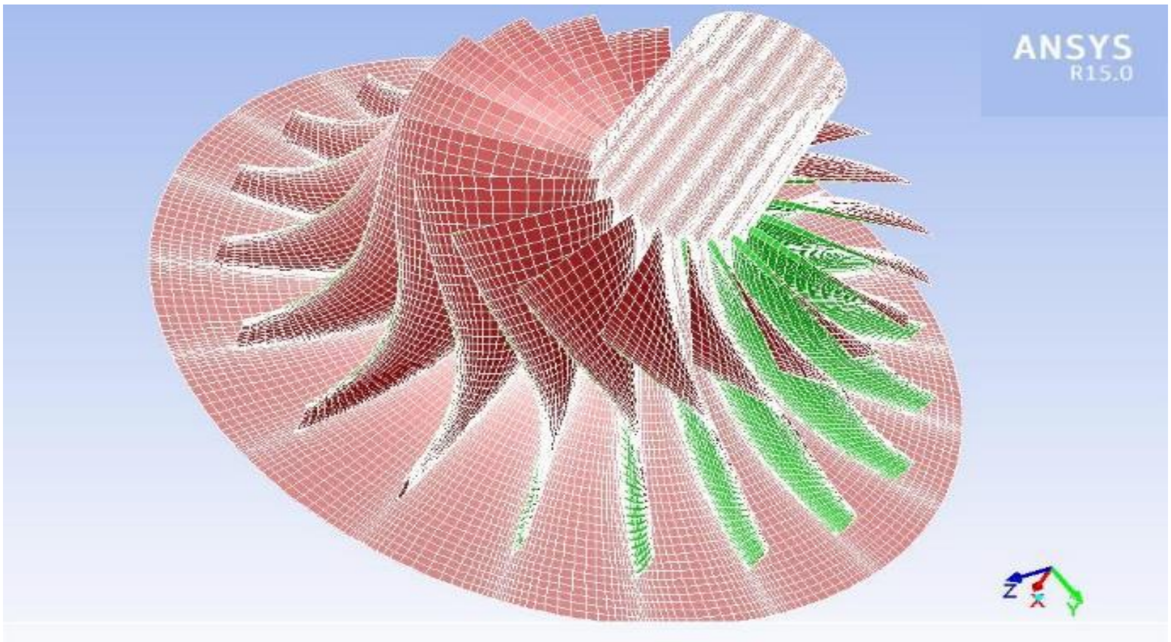


Figure 3.Mesh Model of Compressor Blades

3.2BOUNDARY CONDITIONS, RESULTS AND DISCUSSION

Table 2 Boundary conditions

Hub	Wall
Inlet	Pressure inlet
Interior	Interior
Outlet	Pressure outlet
Periodic wall 1	Periodic
Periodic wall 2	periodic
shroud	Wall
Wall blade pressure	Wall
Wall blade suction	wall

The following are the parameters taken for the analysis of the centrifugal impeller. The walls are adiabatic.

Pressure inlet - 1 bar

Back pressure - 1.59 bar

Temperature inlet - 288.1 K

Isentropic coefficient (γ) =1.4

Gas constant of air, R =287 J/kg/K

Rotational speed, N = 14000 rpm

Overall diameter of impeller = 0.4 m

Eye tip diameter = 0.28 m

Eye root diameter = 0.09 m

Air mass flow, $\dot{m}$ = 5.4 kg/s

Inlet stagnation temperature, T01 = 288 K

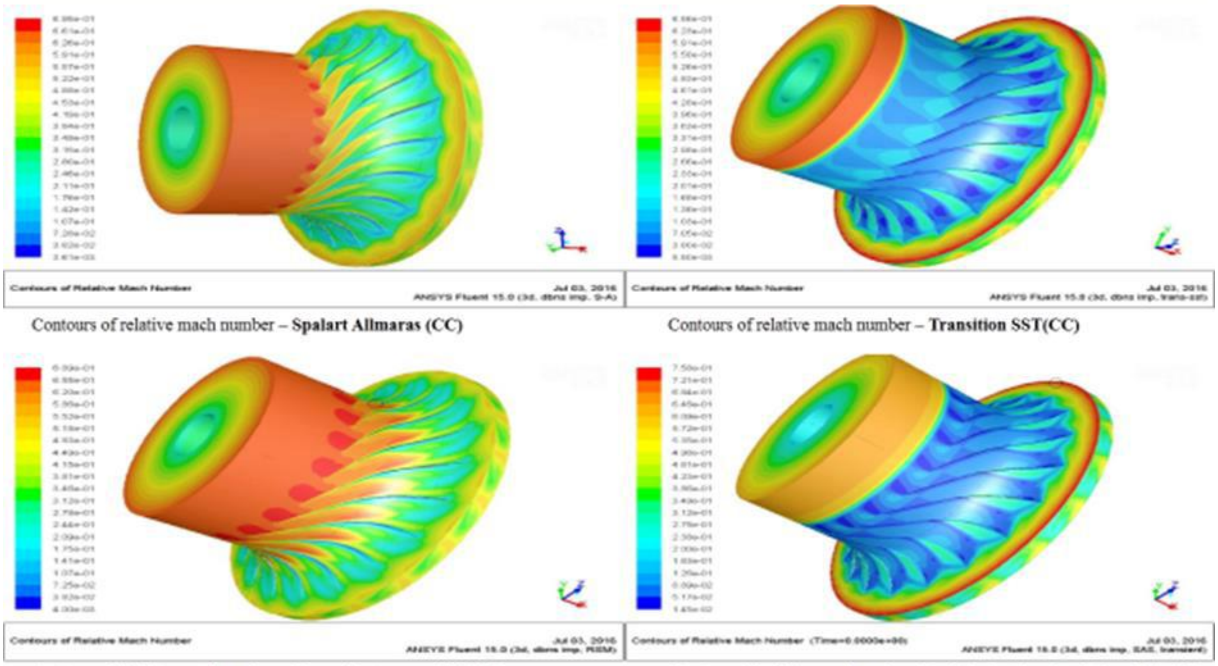
Inlet aero dynamical blockage factor, B1 =0.02

Inlet stagnation pressure, p01 = 1 bar

Absolute inlet flow angle, $\alpha = 0\times\pi/180 = 0^0$.

After the solution iterations are initialized the following are the results obtained

17th February 2018 www.conferenceworld.in ISBN: 978-93-87793-01-9



Contours of relative mach number – **Spalart Allmaras (CC)**

Contours of relative mach number – **Transition SST(CC)**

Contours of relative mach number – **Reynolds stress model (CE)**

Contours of relative mach number- **Scaled adaptive simulations(CC+CE)**

Figure 4 Contours of relative mach number for Spalart Allmaras, Transition SST, Reynolds stress model, Scaled Adaptive Simulation.

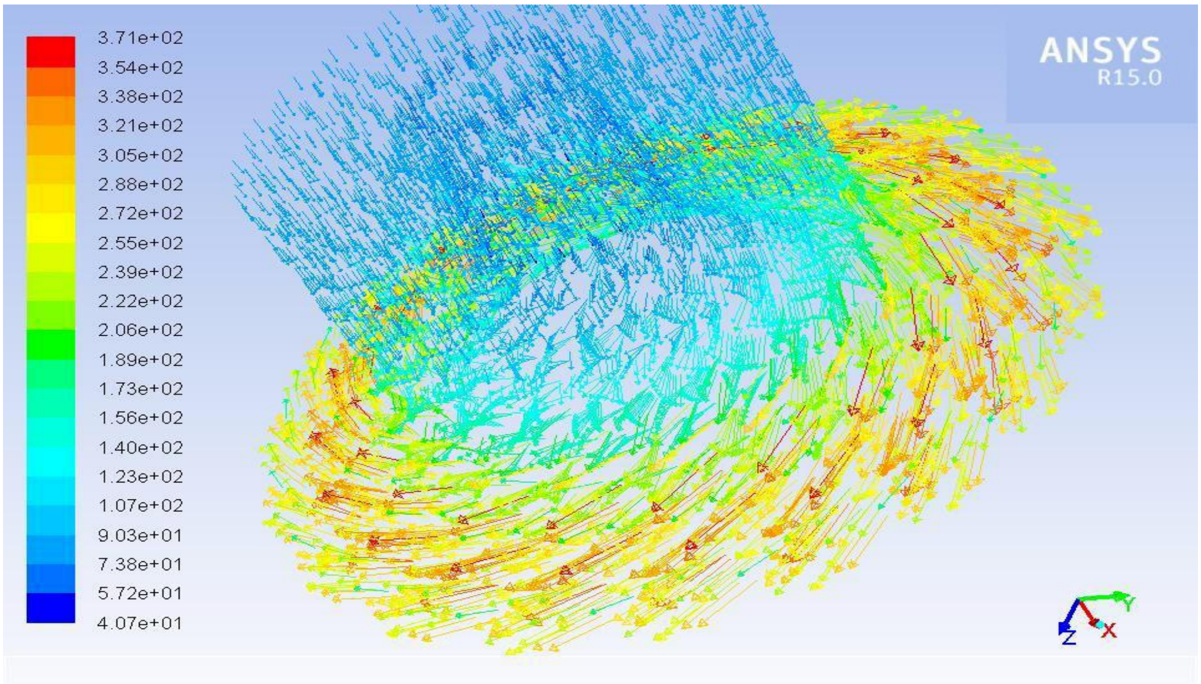


Figure 5 Streamlines through Impeller

Table 3: Discharge, Pressure ratio vs Speed @1.6 bar back pressure

S.no	Speed(rpm)	Mass flow rate(kg/s)	Pressure ratio(p_d/p_s)
1	14000	0.27	2.20
2	14500	0.32	2.29
3	15000	0.35	2.39
4	16000	0.4	2.64
5	16300	0.4	2.65
6	16400	0.4	2.65
7	16500	0.4	2.65

Table 4 Back pressure vs Discharge, Pressure ratio @ 15000 rpm

S.no	Back Pressure (Bar)	Mass flow rate (kg/s)	Pressure ratio (p_d/p_s)
1	1.6	0.38	2.35
2	1.7	0.35	2.39
3	1.8	0.29	2.44
4	1.9	0	2.13

Table 5Speed vs Relative Mach number, Discharge and Pressure ratio.

SL.NO	Speed	Relative Mach Number	Mass Flow Rate (Kg/s)	Pressure Ratio (P_d / P_s)
1	14000	0.689	0.27	2.20
2	15000	0.957	0.35	2.39
3	16000	1	0.4	2.64
4	16500	1.09	0.4	2.64

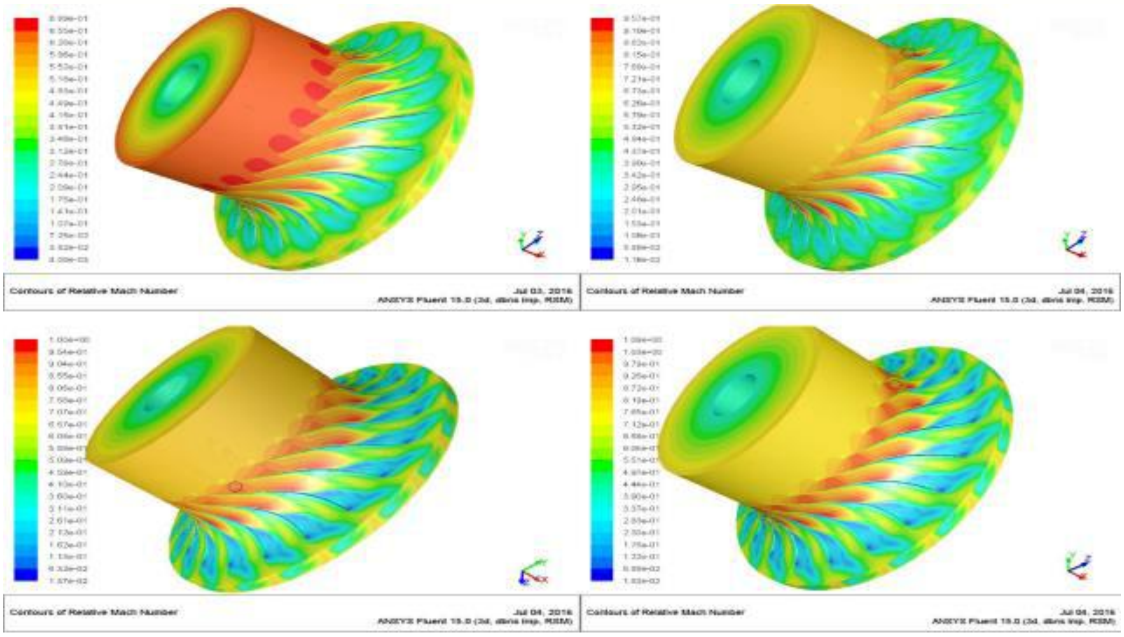


Figure 6 Contours of Relative Mach number at 14000, 15000, 16000, 16500 rpm.

IV.CONCLUSION

The Eckardt centrifugal compressor impeller was considered in the present studyfor investigating the effect of choosing various turbulence models viz Spalart-Allmaras(Curvature Correction),Transition SST (Curvature Correction), Reynolds stress model(Compressibility Effect), Scaled Adaptive Simulations (Curvature Correction +Compressibility Effect) present in fluent software on performance prediction ofcompressor. It is observed that values of various parameters total pressure, totaltemperature, blade angles, torque, isentropic and polytrophic efficiencies etc. (except formass flow) are within range. However due to strong channelled curvature and intensiverotation coupled with high Mach number fluid flow RSM (CE) were considered to besuperior for the problem being investigated.Surge and Choke phenomenon were also investigated in the present study surgegenerally initiated when discharge becomes almost equal to zero and surging causessevere vibrations coupled with reverse flow. Generally the discharge varies with backpressure at centrifugal outlet. It is observed that discharge has almost decreased to zerovalue at a back pressure of 1.8 bar indicating onset of surge phenomena at 1.8 bar backpressure, 15000 rpm. Hence to avoid surge it would be necessary to ensure that backpressure does not increase beyond 1.6 bar. Choke phenomena arise when dischargereaches maximum value remains constant when speed is increased further. For thecompressor under study the discharge was found to increase from 0.27 kg/s to 0.4 kg/s(per blade) when speed is increased from 14000 rpm to 16000 rpm. The maximum valueof 0.4 kg/s remained constant even when speed is increased from 16000 rpm to 16500rpm indicating onset of choking phenomena at 16000 rpm.

REFERENCES

[1.] Michael L. Shur, Michael K. Strelets, and Andrey K. Travin ,Philippe R. Spalart , "Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart–Shur Correction", Copyright ©2000 by the American Institute of Aeronautics and Astronautics, AIAA JOURNAL Vol. 38, No. 5, May 2000.

[2.] Pavel E. Smirnov, Florian R. Menter , "Sensitization of the SST Turbulence Model to Rotation and Curvature by Applying the Spalart–Shur Correction Term", JOURNAL OF TURBOMACHINERY, Copyright © 2009 by ASME OCTOBER 2009, Vol. 131 / 041010-1, [DOI: 10.1115/1.3070573]

[3.] K. J. Elliott, E. Savory, C. Zhang, R. J. Martinuzzi and W. E. Lin, "Analysis of a curvature corrected turbulence model using a 90 degree curved geometry modelled after a centrifugal compressor impeller".

[4.] F.R. Menter, M. Kuntz and R. Bender, "A Scale-Adaptive Simulation Model for Turbulent Flow Predictions",Copyright © 2003 by CFX. Published by the American Institute of aeronautics and Astronautics, Inc AIAA 2003-767.

[5.] Luca Mangani, Ernesto Casartelli, Sebastiano Mauri, "Assessment of Various Turbulence Models in a High Pressure Ratio Centrifugal Compressor with an Object Oriented CFD Code", JOURNAL OF TURBOMACHINERY. Copyright © 2012 by ASME NOVEMBER 2012, Vol. 134 / 061033-1

[6.] Jie Li, Yuting Yin, Shuqi Li and Jizhong Zhang, "Numerical simulation investigation on centrifugal compressor performance of turbocharger",Journal of Mechanical Science and Technology 27 (6) (2013) 1597~1601,DOI 10.1007/s12206-013-0405-3

[7.] www.springerlink.com/content/1738-494x.

[8.] Shalini Bhardwaj, Dr. K. K.Gupta, "Centrifugal Compressor Analysis by CFD", International Journal of Science and Research (IJSR), Volume 3 Issue 11, November 2014

[9.] P. Le Sausse, P. Fabrie, D. Arnou, and F. Clunet, "CFD comparison with centrifugal compressor measurements on a wide operating range", EPJ Web of Conferences 45 01059 (2013),DOI: 10.1051/ epjconf / 20134501059, © 2013 by EDP Sciences, 2013

[10.] Ing. Martin BABÁK, Ph.D, "CFD analysis of a surge suppression device for high pressure ratio centrifugal compressor", ANSYS konference 2010, Frymburk 6. - 8. října 2010

[11.] Yu Wang, Zhen Luo, "Simulation and Performance Analysis on Centrifugal Compressors of Different Dimensions and Variable Operation Speed", 978-1-61284-459-6/11/$26.00 © 2011 by IEEE.

[12.] R. A. Tough, A. M. Tousi, J. Ghaffari, "Improving of the micro-turbine's centrifugal impeller performance by changing the blade angles", Copyright © 2010 ICCES, vol.14, no.1, pp.1-22.

[13.] G. Sravan Kumar, Dr. D. Azad and K. Mohan Laxmi, Valuation of Turbulence Modelling on Low Speed Centrifugal Compressor Using Computational Fluid Dynamics. International Journal of Mechanical Engineering and Technology, 7(6), 2016, pp. 634–641.

[14.] S.M. Swamy, V. Panndurangadu and J.M. Shamkumar. Effect of a Tip Clearance on the Performance of a Low Speed Centrifugal Compressor. International Journal of Mechanical Engineering and Technology, 8(1), 2017, pp. 178–188.