Scientific Journal of Impact Factor (SJIF): 4.72
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 4, Issue 10, October -2017

Computational Fluid Dynamics Analysis for Centrifugal Compressor using Advanced Technique

Bharatkumar Manharlal Trivedi [1], Dr Jeetendra A. Vadher[2]

[1].Research Scholar Rai University, Ahmedabad.
[2] Professor and head, Mechanical Engineering Department, Government Engineering College, Palanpur

Abstract — *To obtain more power from the engine, a new and the larger turbocharger is being used. This paper is the study of the complete fundamental study of air flow physics. Turbochargers are mostly used throughout the automobile industries as they can enhance the output of an internal combustion (IC) engine without the need to increase its cylinder capacity. This paper deals with the computational fluids dynamics analysis of flow in high speed turbocharger. The. These characteristics derived from Finite Element Analysis and then it will be compared with experimental results.*

Keywords- *Centrifugal Compressor, Turbochargers, FEA, CFD, IC Engine.*

1.INTRODUCTION

Computational Fluid Dynamics constitutes a new "3rd approach" in the philosophical study and development of the whole discipline of fluid dynamics. In the Seventeenth century, the foundations for experimental fluid dynamics were laid. The Eighteenth and Nineteenth centuries saw the gradual development of theoretical fluid dynamics. As a result, throughout most of the Twentieth century, the study and practice of fluid dynamics involved the use of pure theory on the one hand and pure experiment on the other. The learning of fluid dynamics as recently as, say, 1960, involved operating in the "2-approach world" of theory and experiment. However, the advent of the high speed digital computer combined with the development of accurate numerical algorithms for solving physical problems on these computers has revolutionized the way we study and practice fluid dynamics today. It introduced a fundamentally important new 3rd approach in fluid dynamics – the approach of CFD. Computational Fluid Dynamics (CFD) is a latest tool of fluid analysis software predicts the interaction of a working fluid with its geometrical surroundings and operational environment. Accurately predicting these interactions is highly dependent on understanding the energy loss models embedded within the design code. These loss models dictate how severely performance diminishes due to inherent or sometimes improper geometrical and operational constraints. Such energy losses include skin friction, excessive pressure recovery, airfoil incidence, flow recirculation, and blade tip leakage to name a few.

2. Description of Test Rig and Instrumentation:

In order to obtain useful results a very simple system was chosen for the experimental study. The test rig is shown in fig.2.1

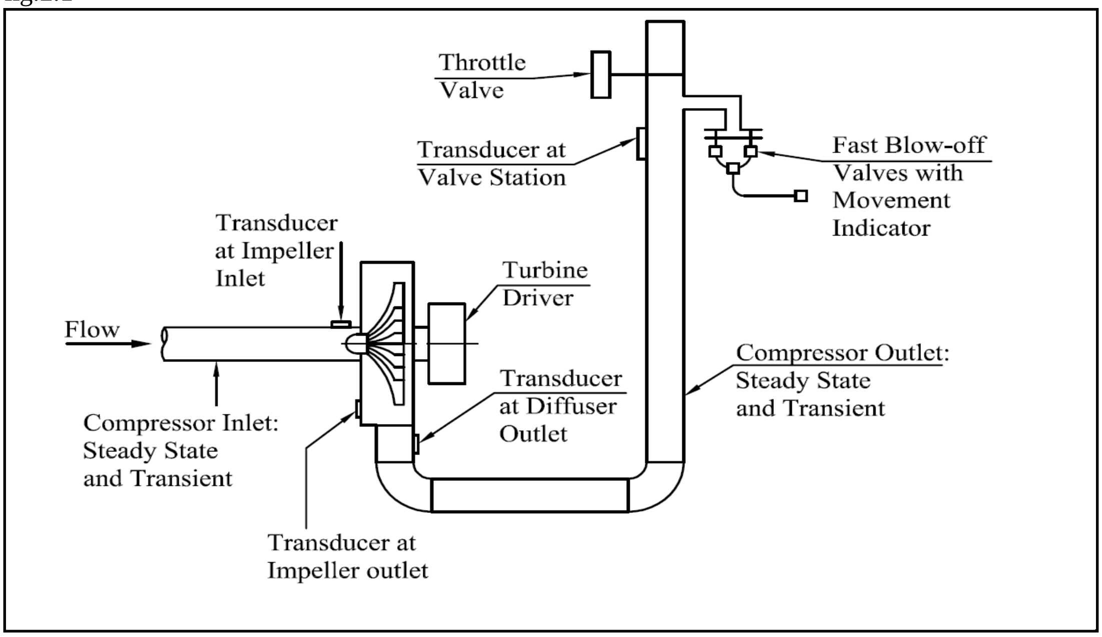

Figure 2.1 Compressor transient rig layout

The compressor was a small (200mm impeller diameter), single stage, centrifugal compressor. The compressor speed selected for the test was 90,000 rpm. The characteristic of the compressor at the chosen speed is shown in fig. 3.9. These characteristics derived from Finite Element Analysis and then it will be compared with experimental results.

The duct work upstream of the compressor consisted of a straight pipe and an inlet bell mouth. The downstream ducting had two right angle bends and a branch to which a fast acting blow-off valve was mounted. (The valve stroke times for the opening and closing of the blow-off valve were 0.25 seconds. and 0.5 seconds respectively.) The blow-off valve was a repeatable poppet actuator type, and fast transients were introduced into the system by the operation of this valve. The compressor delivery air exhausted through a throttle valve, and the compressor steady state operating point on its characteristic was determined by the actuation of this throttle valve.

High performance pressure transducers were mounted at the system inlet, impeller inlet, and compressor outlet and blow-off valve stations respectively. Due to the relative small changes in pressure ratios during the transients considered and the slowness of the temperature changes the effect of temperature on the pressure readings were neglected.

The duct work upstream of the turbine section was a straight pipe via a combustion chamber connecting to the high pressure air supply. The exhaust from the turbine was discharged via a straight vertical pipe to the ambient. The combustion chamber becomes operational if insufficient energy is available in the high pressure air (cold) driving the compressor at the required speed and pressure ratio (preheating using the combustion chamber was required for operating conditions above 60,000 rpm).

3. Finite Element Analysis and Characteristic Curve:

Computational Fluid Dynamics (CFD) predicts the interaction of a working fluid with its geometrical surroundings and operational environment. Accuracy of predicting these interactions of fluid in the body is completely dependent on understanding the energy loss models which is co-related within the design code. These loss models decide how severely performance reduces due to inherent or sometimes improper geometrical and operational constraints. Such energy losses include surface friction, excessive pressure drop, airfoil phenomena, recirculation of flow, and blade tip leakage to name a few.

The compressor has 24 blades that revolve about the Z-axis. A clearance gap exists between the blades and the shroud of the compressor. The outer diameter of the blade row is approximately 40 cm.

To run analysis of the aerodynamic performance, a mesh shall be created in both ANSYS Turbo Grid and the Mechanical application using an existing design which is to be reviewed beforehand in ANSYS Blade Gen. Once the meshes have been created, initial parameters defining the aerodynamic simulation will be set in CFX-Pre and then solved in CFX-Solver. The aerodynamic solution from the solver will then be processed and displayed in CFD-Post.

A mesh can be created in both ANSYS Turbo Grid and the Mechanical application. Turbo Grid will create a CFD mesh that will be part of the fluid domain. The Mechanical application will generate the solid blade mesh that is required for solving for the volumetric properties in the blade.

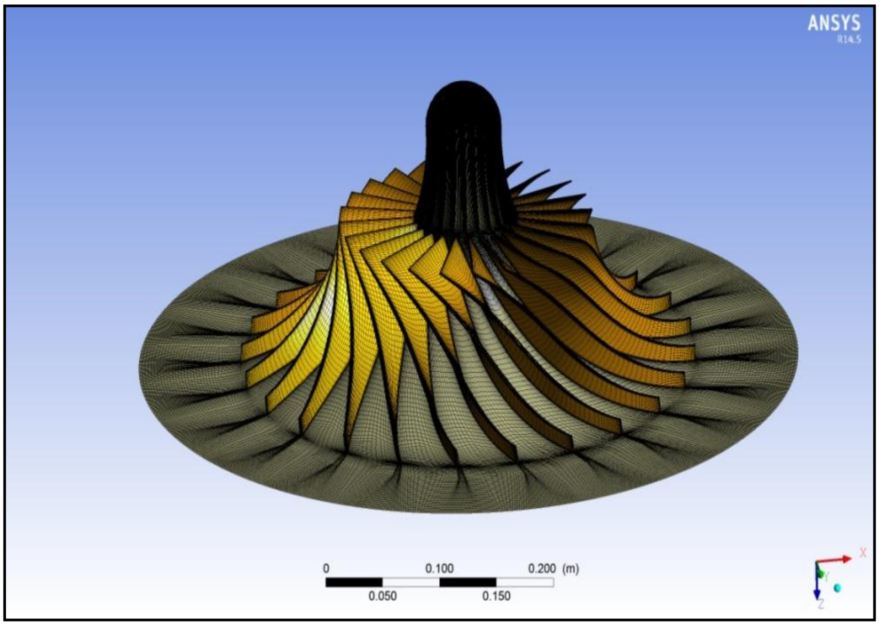

Figure 3.1 Mesh generation of rotor of the compressor

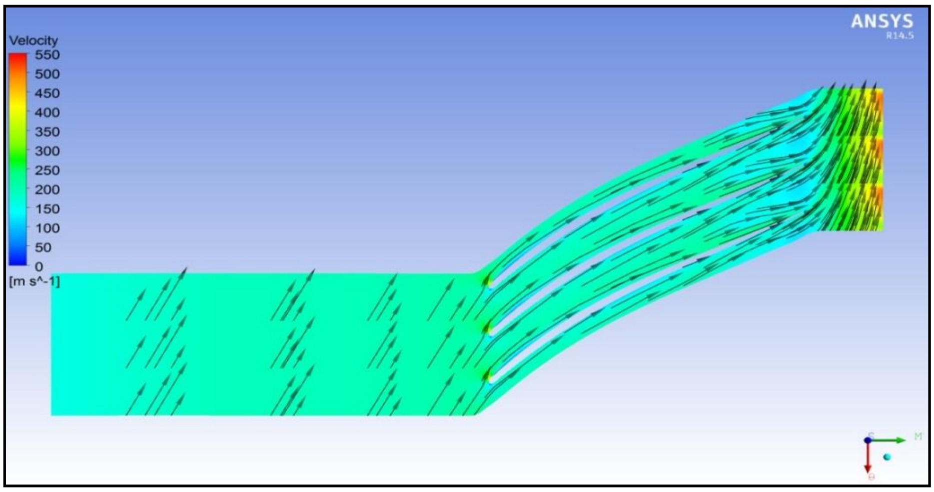

Figure 3.2 Velocity distribution

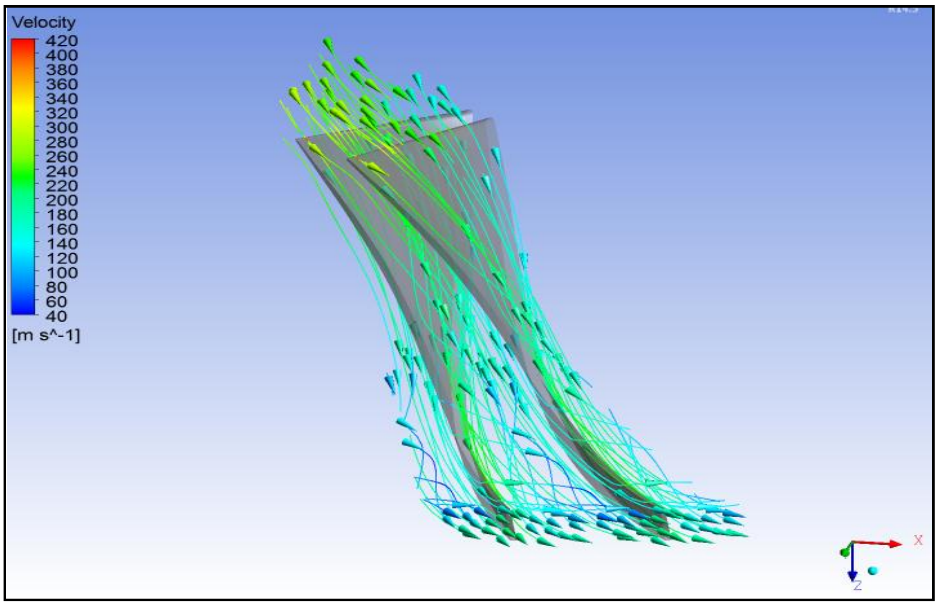

Figure 3.3 Stream Line view

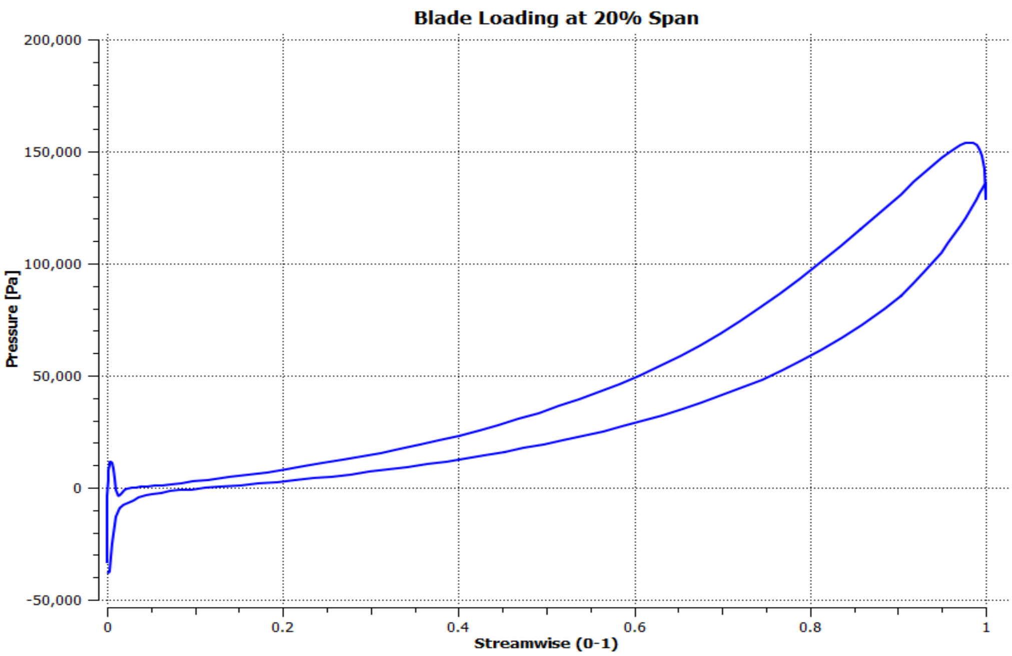

Figure 3.4 Pressure distribution at 20% span of blade

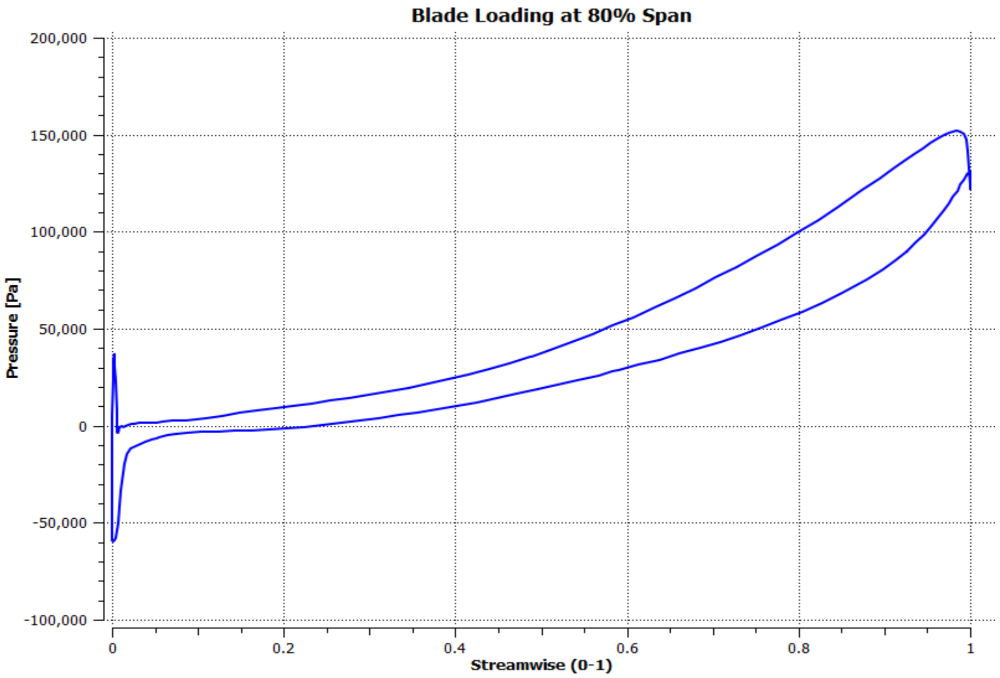

Figure 3.5 Pressure distribution at 80% span of blade

After performing successive analysis for various interval of flow rate value, the continuous curve of Polytropic head can be found out at specific speed and it is plotted in figure 3.6 which is called Characteristics curve of compressor

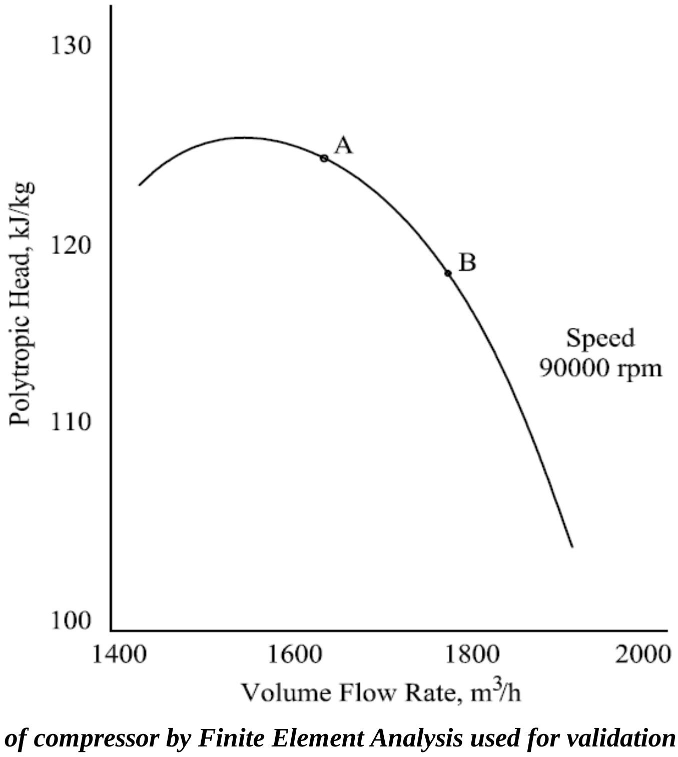

Figure 3.6 Characteristics of compressor by Finite Element Analysis used for validation of mathematical model (Point A and B achieved by experimentally)

.

3. Compressor Transients between Stable Points

By actuation of the blow-off valve a fast transient was introduced into the system. The repeated opening and closing of the blow-off valve transients between the corresponding two stable points can be obtained. Sufficient time was given between the opening and closing of the valve so that the post transient steady state operating point can be obtained. During the transient the compressor speed remained nominally constant. The stable points due to the opened and closed position of the blow-off valve are shown by point A and B respectively in above figure.

4. Conclusion

Surge test will be performed by adjusting the flow through the throttle valve. From the above work it is concluded that the transient due to the closing of the blow-off valve was sufficient to take the compressor into surge and the compressor will be made to operate at point A on the characteristic curve.

REFERENCES

1. Davis, F. T., Corripio, A. B., "Dynamic Simulation of Compressor using a Real Gas Equation of State."
2. Dean, R. C., Young, L. R., "The Time Domain of Centrifugal Compressor and Pump Stability, Stall and Surge." March 1976.
3. Rothe, P. H. Runstadler, P. W. First Pump Surge Behaviour. J. Fluid Eng. Dec. 1978. p. 439-466
4. Fasol, Kitt, Tuip, L., Identification of Blast Furnace Air Supply System. Paper 8.6,4th IFAC Symposium on Identification and System Parameter Estimation, TbiUsi, Sept. 1976.
5. Fasol, Kitt, Gironau, M., Tuip, L. "Possibilities of Theoretical and Experimental Identification of Blast Furnace Air Supply System," VDI Berichte, 276, 1977, p. 18-85.
6. Fasol, K. H. Gironau, M., "Modelling and Simulation of a Chemical Process Plant Blast Furnace System." IMACS Symposium on Simulation of Control Systems, Vienna, Sept. 1978.
7. Kulberg, J. F., Sheppard, D. E., King, E. O., Baker, J. R., "The Dynamic Simulation of Turbine Engine Compressors." AIAA 69-486, AIAA 5th Propulsion Conference.
8. Willoh, R. G. Seldner, K.Multistage Compressor Simulation Applied to the Prediction of Axial Flow Instabilities. NASA TMX-1880. Sept. 1969.
9. Corbett, A. G., Elder, R. L., Stability of an Axial Flow Compressor with Steady Inlet Condition. J. Mech. Eng. Sci.Vol. 16, No. 6, 1975.
10. Corbett, A. G., Elder, R. L.," Mathematical Modelling of Compressor Stability in Steady and Unsteady Flow Conditions.Unsteady Phenomena in Turbo machinery,
11. Elder, R. L., Macdougal, I., "Simulation of Centrifugal Compressor Transient Performance for Process Plant Application", ASME paper, 1983.
12. Greitzer, E. M., "Surge and Rotating Stall in Axial Flow Compressor Part II - Experimental Results and Comparisons with Theory", J. Eng. For Power, April, 1978, p. 199.
13. Schultz, J. M., "The Polytropic Analysis of Centrifugal Compressors. J. Eng. for Power, Jan 1962, p. 62-82."
14. Royal Aero. Soc... "Friction Losses for Fully Developed Flow in Straight Pipes", Engineering, Sci. Data Sheet 66027,1966.
15. Royal Aero. Soc., Pressure Losses in Curved Ducts: Single Bends. Eng. Sci. Data Sheet 77008,1977.
16. A. Eli Nisenfeld, "Centrifugal Compressor Principals of Operation and Control. ISA 67, Alexander Drive, P. O. Box 12277, Research Triangle Park,. NC 27706.
17. Curtis F. Gerald, Patrick 0. Wheatley, "Applied Numerical Analysis" Third Edition, Addison Wesley 11577
18. Elder, R. L., "Mathematical Modelling of Axial Compressors. Ph. D. Thesis, Leicester University, 1972."
19. Kemp. U., "The Control of Turbo-Compressor. Sulzer Technical Report", 1963, Vol. 45, part 2
20. Roberts, P. D., Methods for Anti-Surge Control of Centrifugal Compressors. Inst. M. C. Symposium, Turbine and Compressor Control, April 1979.
21. M. Bolemant, D. Peitsch, "An Alternative Compressor Modeling Method within Gas Turbine Performance Simulations", Deutscher Luft- und Raumfahrtkongress, 2014
22. H. Akima, "A new Method of Interpolation and Smooth Curve Fitting Based on Local Procedures", J.ACM, Vol. 17 no 4, 1970
23. A. Weckend, H. Schulte, K.-J. Schmidt, S. Staudacher, "Multistage Compressor Model for Transient Performance Simulation", ASME Turbo Expo 2008.
24. Haiwen Ye, W. Berns, S. Staudacher, "Modelling Multistage Compressors by Several Split Compressors", ISABE, 2003
25. V. Stanciu, E. Popovici, "Centrifugal Compressors Surge Simulation", U.P.B. Sci. Bull., Series D, Vol. 75, Iss. 3, 2013
26. V. Silivestru, R. Petcu, V. Vilag, C. Mandiş, D. Robescu, "Mathematical modeling and Numerical Simulation of process inside a Turbo engine", Scientific Bulletin edited by The University Politehnica of Bucharest, Series D, Vol. 71, Iss. 3, 2009
27. Sanford Fleeter, et.al. "Gas Turbine Engine Compressor-Combustor Dynamics Simulation Design", Computer Science Technical Reports, Purdue University, Paper 1437
28. Tiziano Ghisu, "CFD Analysis of Inlet Flow Distortions on an Axial Fan"
29. Hirai, K., Kodama, H., Nozaki, Kikuchi, Tamura, Matsuo, 1997, "Un-steady Three-Dimensional Analysis of Inlet Distortion in Turbo machinery", AIAA-97/2735
30. Ghisu, T., 2003, "Axial Compressor Response to Inlet Flow Distortion by a CFD Analysis", joint MSc Thesis and Tesi di Laurea, respectively at Cranfield University and Politecnico di Torino
31. Reid, C., 1969, "The Response of Axial Flow Compressor to Intake Flow Distortion", ASME Paper 69-DT-29

32. R.A. Stanley, W.R. Bohannan, “Dynamic Simulation of Centrifugal Compressor Systems”, 6th Turbomachinary Symposium, pp 123.
33. Donald E. Bently, et.al. “Rotor Dynamics of Centrifugal Compressors in Rotating Stall”, p40, ORBIT, 2Q01
34. Wachel, J.C., (1975) “Non-synchronous Instability of Centrifugal Compressors,” ASME 75-Pet-22.
35. Kaneki, T., Eino, T., (1979) “Centrifugal Compressor for Urea Synthesis Plant,” Hitachi Review, Vol. 28, No.6.
36. Sabella, D., Terrinoni, L., Timori, A., (1981) “Full Load Testing of Centrifugal Natural Gas Injection Compressors,” Inst. Mech. Eng. Conference Publications.
37. Bonciani, L., Terrinoni, L., Tesei, A., (1982) “Unsteady Flow Phenomena in Industrial Compressor Stage,” NASA Conference Publication 2250.

