Experimental Investigation of Impeller-Diffuser Interaction in a Centrifugal Compressor

CSME Forum 2006, Calgary

Rita Patel*, Eric Savory*, Robert Martinuzzi**
*Department of Mechanical and Materials Engineering
University of Western Ontario
London, Ontario, Canada N6A 5B9
Phone: 519-854-0152, Fax: 519-661-3030, e-mail: rpatel@uwo.ca

**Department of Mechanical and Manufacturing Engineering
University of Calgary

Abstract
Due to the complex nature of the flow in the impeller-diffuser section of a centrifugal compressor little experimental research has been done in this area. The purpose of the current study is to experimentally investigate the flow field experienced within this section. Laser Doppler Velocimetry (LDV) will be employed to obtain measurements in this highly confined flow field. Before investigation may take place, the LDV technique must be validated for such purpose. This paper presents a novel test facility that is presently being built to model the flow of an impeller by using a stationary cascade rig. In conjunction with CFD results, the test rig will allow for validation of LDV results. Upon full validation, the measurement technique may be applied to a full-scale centrifugal compressor rig.

Acronyms
CFD – Computational Fluid Dynamics
HWA – Hot-Wire Anemometry
LDV – Laser Doppler Velocimetry

Introduction
The centrifugal compressor has a widespread application in both small aircraft engines and helicopters. Consisting of two main components, the rotating impeller and stationary diffuser, its purpose is to achieve the highest possible pressure rise by turning the flow from axial to radial.

Much experimentation has been done on the individual components of the centrifugal compressor. However due to the inaccessibility of the impeller-diffuser section, little work has been done to study the affects of either component on the other. The impeller exhibits a circumferentially non-uniform (jet-wake pattern) at discharge, which convects into the diffuser. The diffuser then experiences disturbances in the form of inlet angle and velocity fluctuations. Depending on geometric and flow conditions, diffuser vanes produce unsteady pressure disturbances affecting the relative impeller flow.

In terms of the impeller flow Eckardt (1976) observed that system rotation and curvature effects suppress the turbulent mixing at the jet-wake shear layer causing the distorted flow to be preserved and enhanced up to the impeller discharge.

As for impeller effects on diffuser performance, Dean and Senoo (1960) used HWA to study the impeller affects on a vaneless diffuser. They concluded that impeller distortions have a great affect on the vaneless diffuser flow behaviour, its extent being dependent on impeller outlet flow angles, degree of distortions and the diffuser wall friction and geometry. Cumpsty and Inoue (1984), using HWA, concurred that circumferentially non-uniform flow is evident in vaneless diffusers even at large radii. They also studied the affects on a vaned diffuser and concluded the following: the unsteadiness of the flow diminishes rapidly with the presence of vanes, the reversed flow back to the impeller is greater than that for vaneless diffuser and the mean wake displacement became lower as the number of vanes increased. The latter affect led to the hypothesis that the mixing of the jet and wake may be stimulated by the presence of vanes.

The previous research mainly looked at the impeller and diffuser performance separately, while Krain (1981), using LDV, analyzed the flow field for an impeller with splitter blade and both vaned and vaneless diffusers. A jet-wake pattern was noticed at the impeller exit (with vaned diffuser), with a further developed wake on the shroud suction side. With a vaneless diffuser similar flow patterns were noticed and Krain concluded that the affect of the diffuser vanes on the impeller flow is negligible for transonic flow into a diffuser. At the vaned diffuser inlet flow angle fluctuations of 17 degrees near the shroud and local velocity fluctuations of 17-20% were noticed due to the impeller discharge pattern. In the diffuser throat, the angle and velocity fluctuations decreased slightly to 13 degrees and 14%, respectively. Gallus et al. (2003a,b) found that smaller radial gaps between the impeller exit and diffuser entrance lead to a more homogeneous flow field at the diffuser vane exit and higher pressure recovery; while impeller efficiency is generally not affected by the radial gap. It was found in Part II of their investigation that with a smaller radial gap more of the impeller wake is convected into the highly loaded diffuser vane pressure side, reducing its loading and leading to better diffusion.

Finally, due to the complex nature of the flow and hard-to-access experimentation areas in full-scale centrifugal compressors, little work as has been done to validate CFD predictions of the flow. The main purpose of this project is to further investigate the impeller discharge and diffuser inlet conditions and their interaction, by use of LDV. In addition, the research will test then validate CFD predictions in full-scale compressor rigs.

This paper will focus on introducing a stationary cascade rig facility to model the behaviour of flow experienced in a centrifugal compressor impeller and describe how the LDV technique will be utilized accurately in such a complex flow field.

Stationary Cascade Rig

Due to the complexity of the flow field in a centrifugal compressor and the great demands of the LDV technique, a novel test facility has been designed. The test facility will allow for validation of the results obtained by LDV measurements and be based on the ability for the LDV system to capture flow phenomena such as separation and secondary flow patterns experienced in previous experimentation.

As well, the ability to optically access the flow in a highly confined volume, especially in such regions as the impeller blade passages and exit will be evaluated.

Finally, in conjunction with CFD simulations the test facility will be used to optimize the particle seeding configuration. The principal parameters to be considered are: the location for introducing particles, the influence of the particle feed pressure on the flow field, optimization of particle size and rate. Of particular concern is the formation of agglomerates and deposition on the optical ports. The CFD simulation will also be used to estimate the required measurement time and simulate the statistical convergence characteristics of the LDV system.

The designed test facility simulates a typical impeller exit flow field in steady state through a non-rotating cascade configuration. The test facility consists of five stationary impeller blades and blade passages. Figure 1 shows a typical impeller that was used to design the cascade model depicted in Figure 2. As the facility is non-rotating, the jet-wake behaviour of a typical impeller is not expected to be seen. However, this facility should still achieve similar quantities of average swirl and Mach number at the exit of a typical rotating impeller.

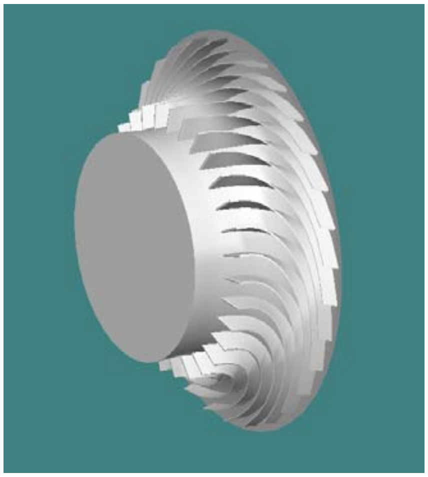

Figure 1:Cascade to Simulate Typical Impeller Outflow Conditions

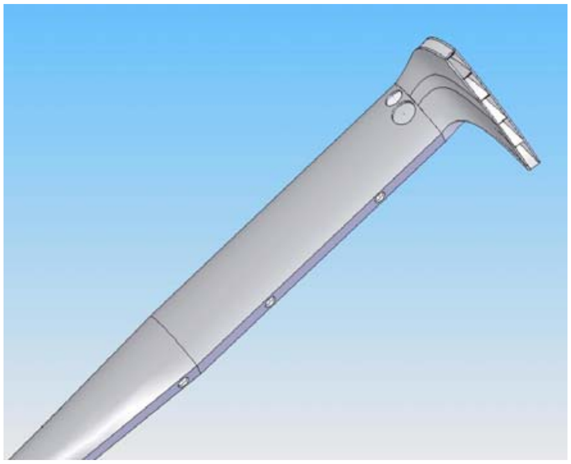

Figure 2: Model of Stationary Cascade Rig Facility

Optical access will be gained by circular glass windows placed upstream of the impeller blades (see Figure 3), and six more will be strategically placed along the 'hub side' of the impeller blade passages (see Figure 4). All optical access windows are removable, thus allowing for fast and easy cleaning if seeding particles cause contamination.

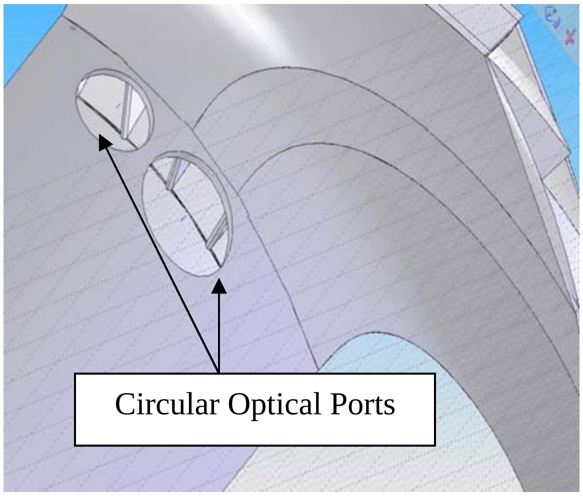

Figure 3: Optical access upstream impeller blades

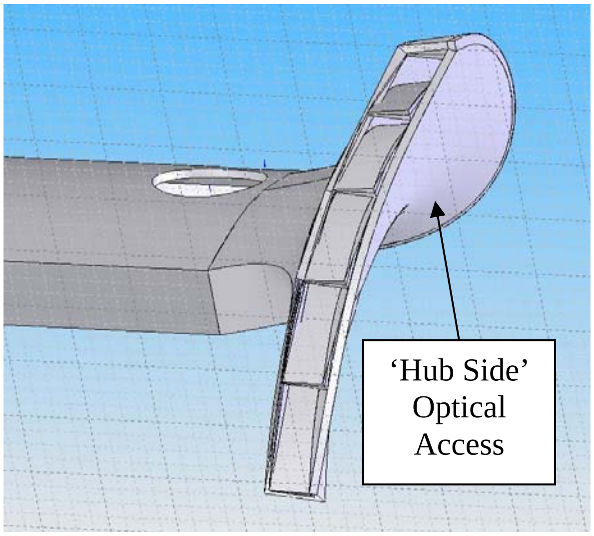

Figure 4: Optical Access on ‘hub side’ of blade passages

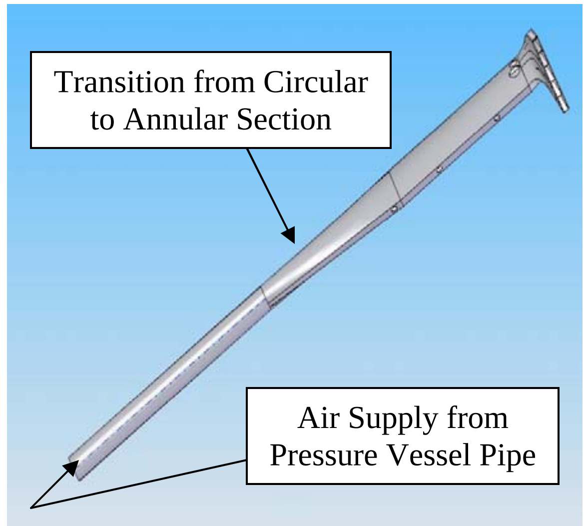

Figure 5: Full View of Stationary Cascade Rig Facility

Air will be supplied to the stationary cascade rig facility, by a pressure vessel discharging air through a 50 mm pipe. Based on a required impeller outlet flow of 290 m/s (M = 0.85) with a swirl angle of 72°, the required pressure of the air supply at the inlet of the facility is calculated to be 175 kPa.

The air supply will flow from the pressure vessel pipe into the facility where it will flow through another 50 mm diameter circular pipe for 0.5 m. The air will then flow through 0.5 m long transition region where the circular cross-section becomes annular, see Figure 5). Finally for another 0.5 m the air will flow through the annular cross section before entering the impeller inlet, and exiting into the atmosphere. Due to the entrainment rate of the jet (Rembold, 2003), a clearance of 0.8 m must be given at the exit of the facility.

The impeller section will be made of Polycarbonate using Fused Deposition Modeling, which provides a tolerance of +/- 0.127mm, while the transition and annular section will be made of sheet metal.

Experimental results will be used validate the LDV technique and discuss the flow phenomena experienced in a stationary impeller and establish if it is at all comparable to that which is experienced in a rotating impeller.

Summary

A test facility to simulate the flow field of the impeller of a centrifugal compressor has been designed. Upon experimentation using the stationary cascade rig facility, seeding characteristics, optical access, and reliability of the results obtained using LDV will be determined and evaluated.

Together with CFD and experimental validation of the LDV technique it may later be implemented on a full-scale centrifugal compressor stage. This will allow valuable flow field data to be obtained to complement existing conventional measurements of temperature and pressure through the compressor stage, thereby aiding the improvement of CFD codes for compressor design.

References

Cumpsty, N. and Inoue, M., (1984), “Experimental Study of Centrifugal Impeller Discharge Flow in Vaneless and Vaned Diffusers,” Journal of Engineering for Gas Turbines and Power, 106, 455-467.

Dean, R. and Senoo, Y., (1960), “Rotating Wakes in Vaneless Diffusers.” Journal of Basic Engineering, 83, 563-574.

Eckardt, D. (1976), “Detailed Flow Investigation Within in a High-Speed Centrifugal Compressor Impeller,” Journal of Fluids Engineering, 98, 390-402.

Gallus, H., Neihuls R. and Ziegler, K., (2003a), “A Study on Impeller-Diffuser Interaction – Part I: Influence on the Performance,” Journal of Turbomachinery, 125, 173-182.

Gallus, H., Neihuls R. and Ziegler, K., (2003b), “A Study on Impeller-Diffuser Interaction – Part II: Detailed Flow Analysis,” Journal of Turbomachinery, 125, 183-192.

Krain, H., (1981), “A Study on Centrifugal Impeller and Diffuser Flow,” Journal of Engineering for Power, 103, 688-697.

Rembold, B., (2003), “Direct and Large-Eddy simulation of Compressible Rectangular Jet Flow,” PhD Dissertation submitted to Swiss Federal Institute of Technology, Zurich.