Journal of Scientific & Industrial Research
Vol. 67, May 2008, pp. 348-354

Flow analysis in centrifugal compressor vaneless diffusers

Ozturk Tatar, Adnan Ozturk and Ali Pinarbasi*

Cumhuriyet University, Eng. Fac. Mech. Eng. Dept,.Sivas, 58140, Turkey

Received 28 May 2007; revised 14 February 2008; accepted 18 February 2008

Flow computations have been performed in vaneless diffuser (diam, 1 m), in low speed backswept centrifugal compressor using finite volume method (FVM), coupled with Standard k-ε turbulent model by solving on commercial package FLUENT assuming steady, incompressible flow condition with design flow-rate in Navier-Stokes equations. Mean velocities and pressure have been computed on eight planes through the diffuser. Flow field at diffuser entry clearly shows impeller jet-wake flow pattern and the blade wakes. Passage wake is located on shroud side of diffuser and mixes out slowly as flow moves through diffuser. On the other hand, blade wakes distort and mix out rapidly in diffuser. Contours of turbulent kinetic energy are also presented on each of the diffuser stations, from which regions of turbulent mixing are identified. Close agreement was observed between numerical simulation and experimental results.

Keywords: CFD, Centrifugal compressor, Diffuser flow, Vaneless diffuser

Introduction

Efficiency of centrifugal compressor depends on flow in impeller and losses in diffuser. In recent years, significant improvements in the performance of centrifugal compressors have been achieved by solving three-dimensional equation for the impeller exit flow. An early development in vaneless diffuser flow calculations[1] was based on calculation of wall friction using radial and tangential momentum equations with continuity equation assuming inlet flow to be uniform and axisymmetric. Dean & Senoo[2] proposed a theory for a discharge mixing process and the concept of a jet/wake flow pattern. Johnston & Dean[3] observed that unusually large total pressure loss at diffuser inlet was due to mixing loss and reversible work exchange. Dean[4] recognised the need to improve diffuser's own efficiency, causing lowering of absolute velocity of the fluid entering to diffuser. Senoo & Kinoshita[5] pointed out that designer is not only concerned with diffuser performance in terms of pressure recovery but also in flow stability diffuser.

Maksoud & Johnson[6,7] investigated three dimensional flow picture and distributions of Reynolds stresses inside vaneless diffuser of a low speed centrifugal compressor with a radial impeller. Hathaway *et al*[8] used a laser anemometer to measure velocities in a low speed centrifugal compressor vaneless diffuser. Mounts & Bras[9] computed unsteady flow field interaction in impeller and vaneless diffuser of a centrifugal compressor using a 3-D Navier-Stokes algorithm showing. Senoo & Nishi[10] investigated flow stability of two dimensional diffusers using a boundary layer calculation, and concluded that time-mean pressure rise in a separated region may be evaluated by assuming a shear stress acting along the edge of separated layer. In spite of these experimental and numerical studies, there is a limited knowledge of diffuser flow and there is no detailed numerical study of diffuser flow phenomenon.

This paper investigates flow effects due to dynamic interaction between flow leaving impeller and diffuser flow regime of centrifugal compressor.

Model Description and Computational Method

Centrifugal compressor surface model was generated using commercial softwares SOLIDWORKS and GAMBIT. Compressor (Fig. 1) has 19 backward curved blades followed by straight wall diffuser with following impeller geometry and operating conditions: inlet blade

*Author for correspondence
E-mail: alipinarbasi@cumhuriyet.edu.tr

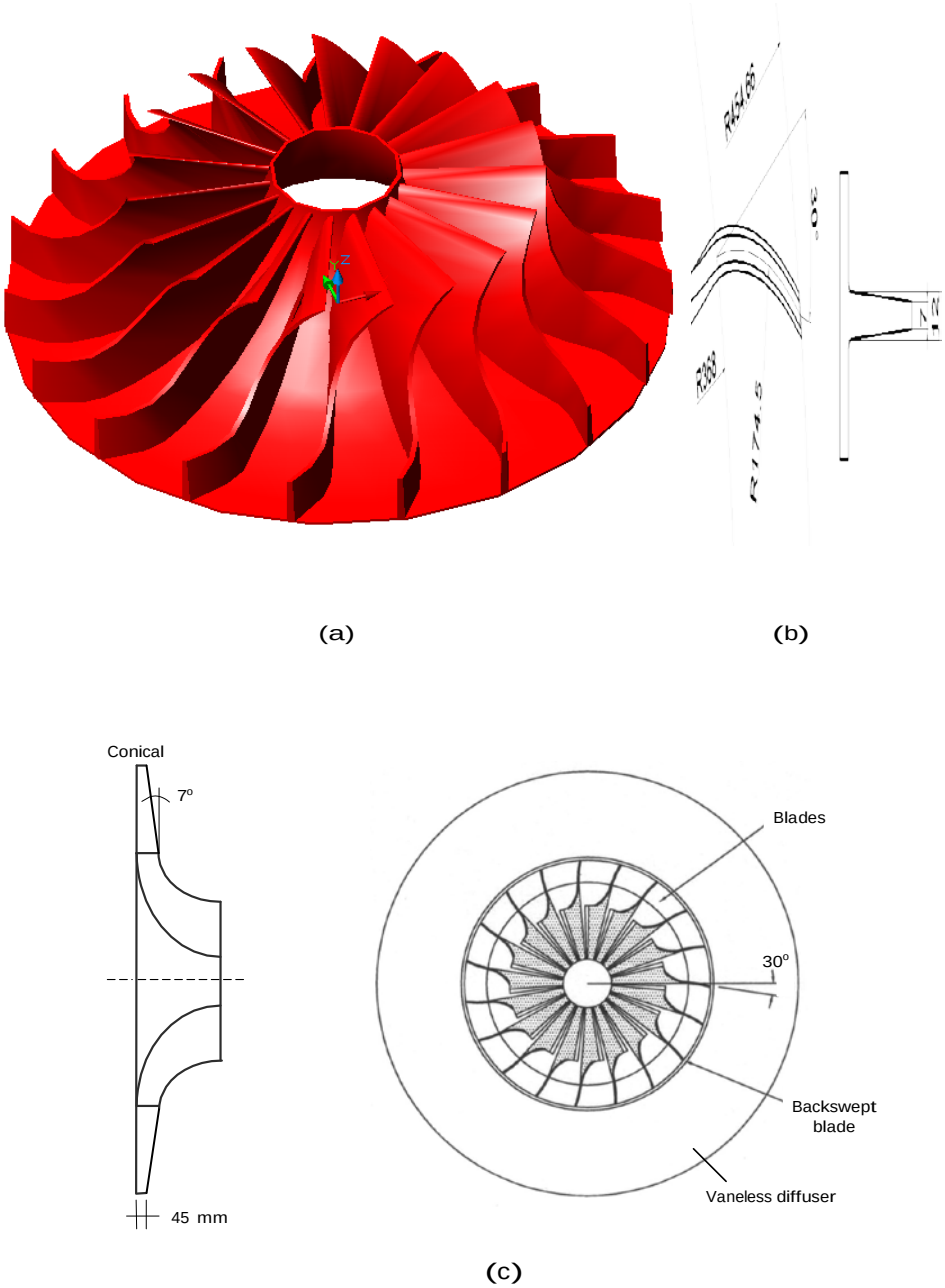


Fig. 1—3-D model and blade details with plane view of compressor

radius at hub (R_h), 88.75; inlet blade radius at shroud (R_s), 283.75; outlet radius (R_o), 454.6 mm; backswept blade angle (²), 30°; outlet blade span (b), 72.3 mm; rotating speed (n), 500 rpm; and flow rate (Q), 0.142 kg/s. Measurement location ratio for different stations (S1-S8) was as follows: S1, 0.02; S2, 0.08; S3, 0.15; S4, 0.21; S5, 0.27; S6, 0.33; S7, 0.39; and S8, 0.45.

In present study, FLUENT was used for calculation with transport equations being solved using Finite Volume Method (FVM). Unstructured hexahedral cells were generated to define inlet (37824 cells) and outlet (14184 cells) zones, while tetrahedral cells were used to define impeller (45420 cells) and diffuser 112118 cells). In total, model has 209546 cells and 153634 nodes with a skewness factor 0.85 (Fig. 2). This size gives correct values by comparing measured 3-D velocities and allows analyzing details of main flow phenomena involved. Diffuser inlet section was 12 mm downstream at impeller.

Code solves Reynolds averaged Navier-Stokes equation in a primitive variable form. Effects of turbulence were modeled using standard k-ε turbulence

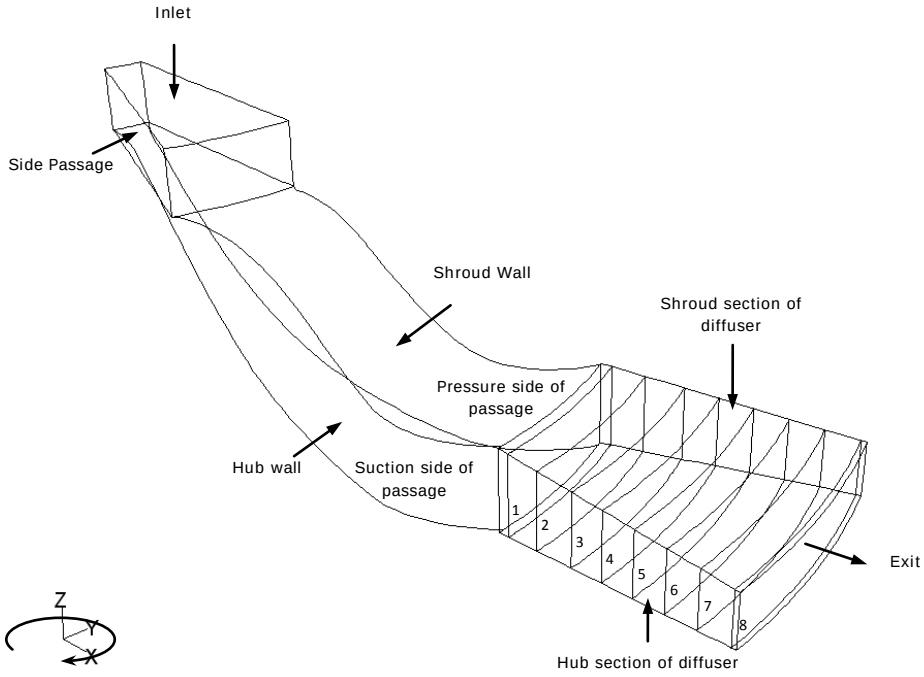


Fig. 2—Computational location of vaneless diffuser

model. Boundary conditions selected are modeled to have more physical meaning for turbomachinery flow simulations, for constant static pressure at the outlet and a variable flow rate proportional to the kinetic energy at the inlet to impeller. Non-slip boundary conditions were imposed over diffuser walls.

Rig and Operating Condition

Numerical work was carried out on low speed centrifugal compressor rig with De Havilland Ghost impeller having 30° backswept blading. Johnson & Moore[11] gave geometry of original impeller. Vaneless diffuser has straight walls and a constant cross sectional area.

Results and Discussion

Mean Velocities

At design mass flow (m = 0.142 kg/s), inlet radial velocity, relatively non-uniform, compared with diffuser suction, indicating that boundary layers symmetrically developed on both walls (Fig. 3). This is in agreement with Pinarbasi & Johnson[12,14] experimental observation. Strong secondary flows were observed in impeller strip boundary layer fluid from the walls and carried in to suction side shroud corner region in a passage wake. At impeller exit, this passage wake was located close to the centre of shroud wall and it moves towards pressure sidewall (Fig. 3). Flow was dominated by a passage vortex rotating in opposite direction of impeller. Developed vortex was strongest in axial to radial bend decaying towards outlet. Vortex formed at blade location was responsible for stabilizing shroud boundary layer, reducing large losses associated with the separation of shroud layer in radial impellers. At the outlet, wake located on shroud tends to be spread more evenly across the shroud as in the case of radial machine. In addition to these features observed in the impeller, a blade wake can also be seen on left hand side of the diagram. The velocity decreased up to 35% within this wake.

Blade wake has been swept to the right by strong tangential velocity component flowing at S 2. Blade wake observed in S 1 (Fig. 3) leaves right hand side of the diagram whilst blade wake from adjacent blade enters to left hand side. Velocity in blade wake was also greatly diminished, indicating rapid mixing out in this region, attributed to strong secondary velocity components existing either side of blade wake, enabling significant increase in shearing flow of wake.

Passage wake was formed along shroud wall by tangential velocities from left to right hand side. Strength of passage vortex was diminished by 60% as driving forces for vortex cease outside impeller. However, wake's shape has been modified as low energy fluid spread further along shroud surface. Mixing out of

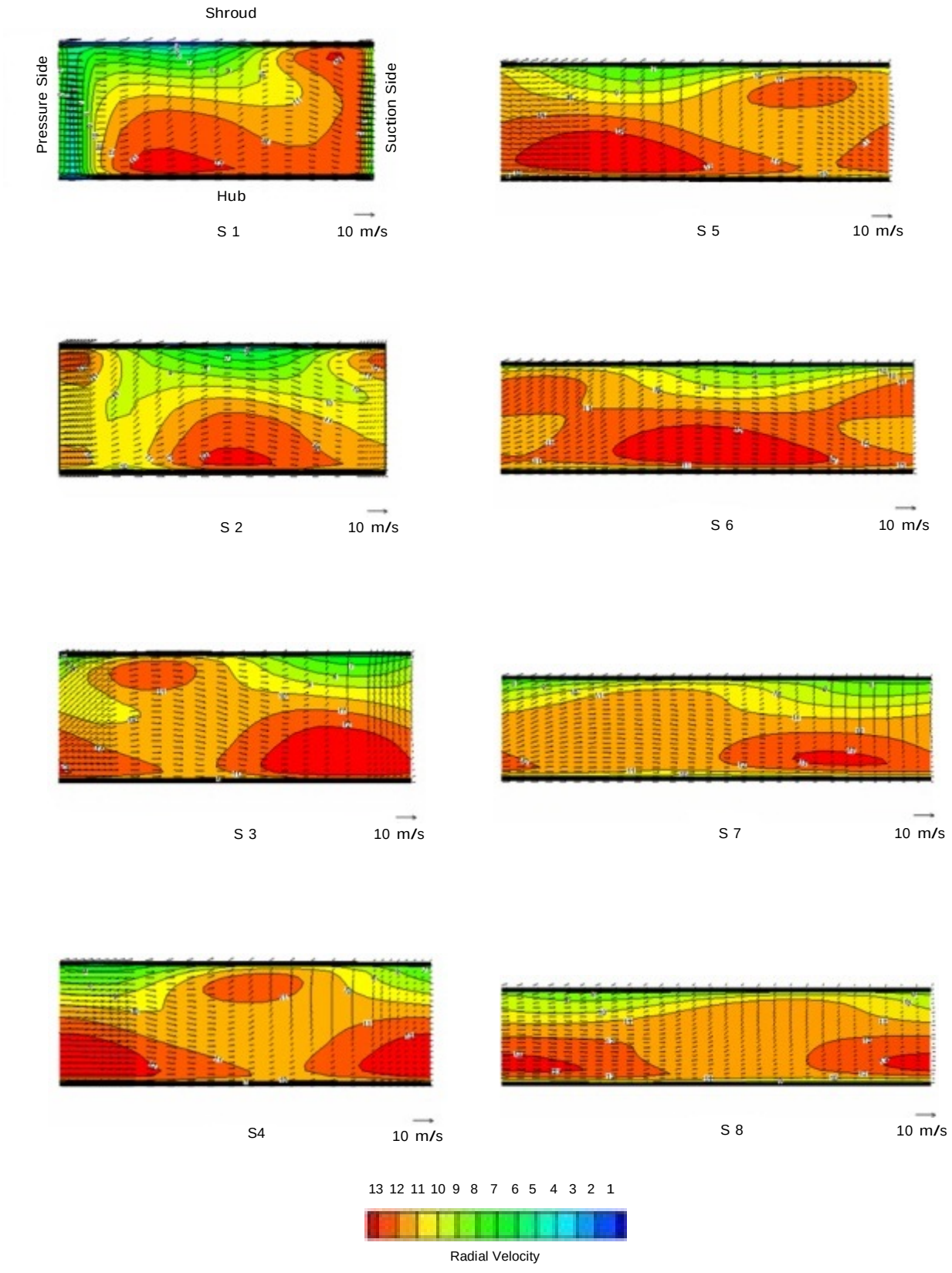


Fig. 3—Radial velocities at vaneless diffuser of centrifugal compressor

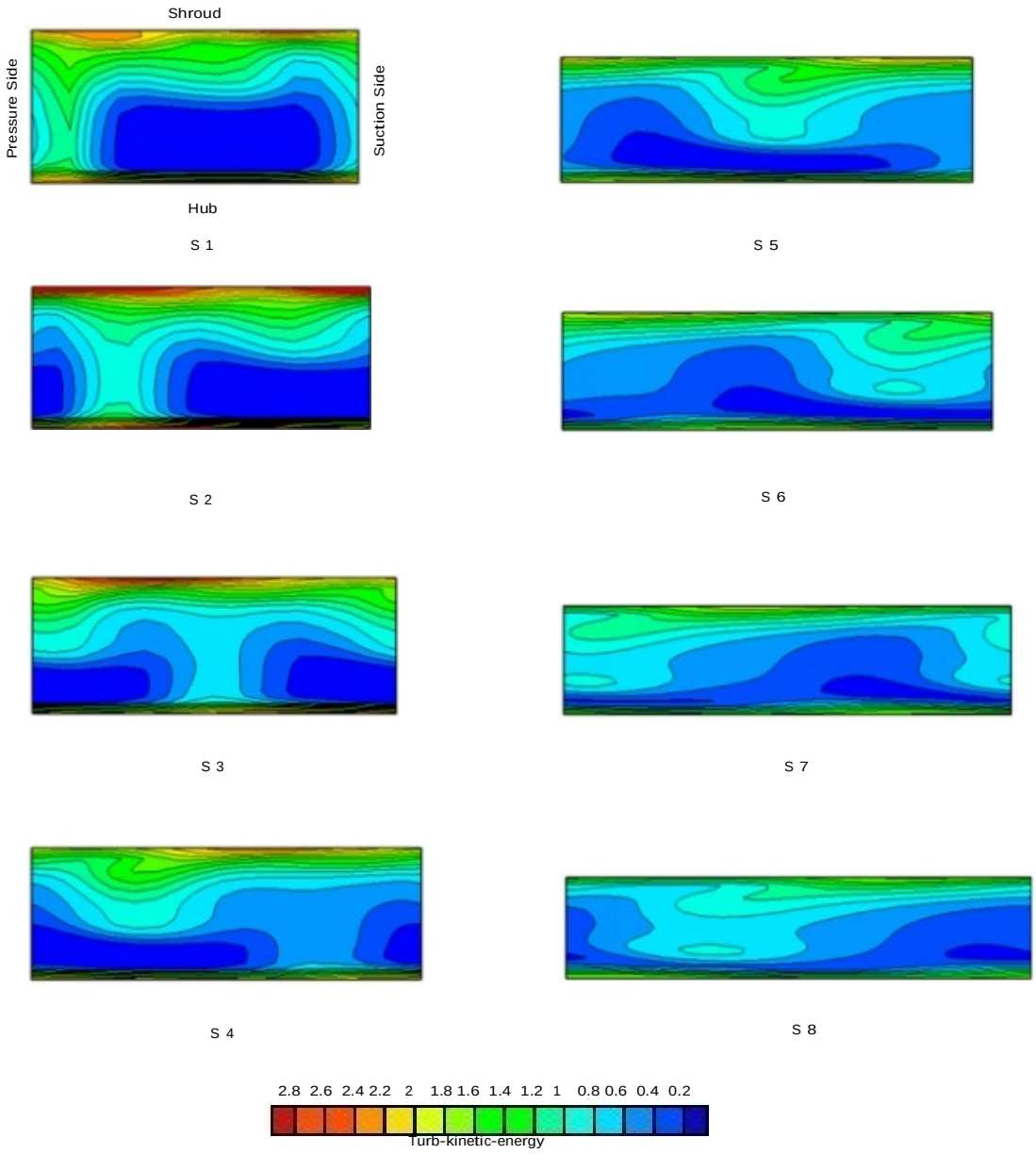


Fig. 4—Turbulent kinetic energies at vaneless diffuser of centrifugal compressor

passage wake was not significantly enhanced, unlike radial impellers running at below design flow rates[15]. For S 2, blade wake swept to the right of passage by secondary velocities. Blade wake mixed out very rapidly at impeller exit, but non-uniformities were still apparent. This was similar to reported[6] movement of blade wake in circumferential direction.

As the flow progresses through S3 to S8 (Fig. 3), variations in blade-to-blade direction were gradually mixed out, and passage wake eventually spreaded out evenly covering whole of shroud surface. Hub wall boundary layer also developed slowly. It was still thinner than shroud boundary layer at S 8. Beyond S 3, flow continued to tend towards a Pouseille flow between two

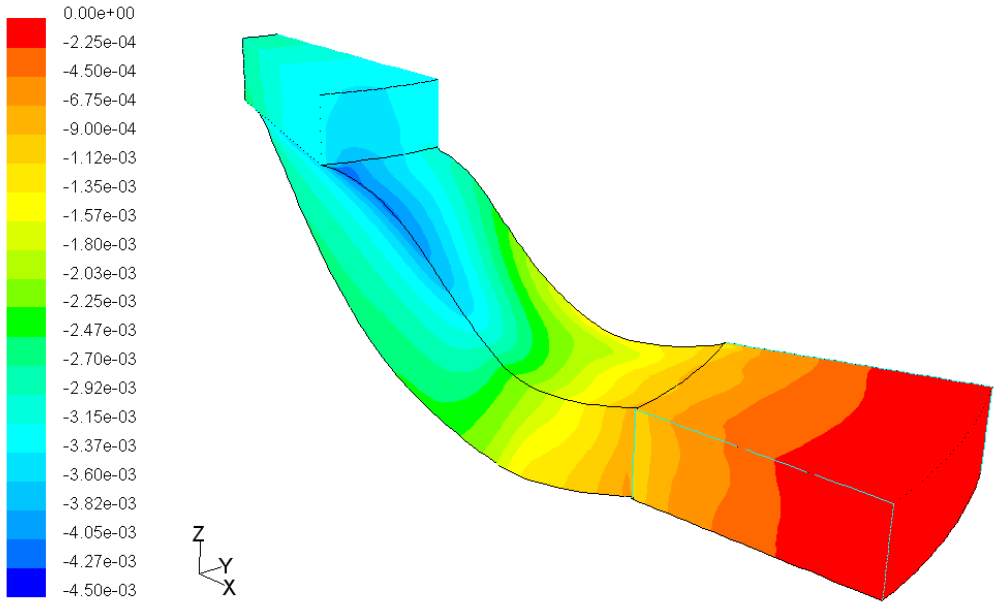


Fig. 5—Contours of total pressure (atm)

diffuser walls as shown by mean velocities at S 8. Passage wake was no longer discernible, although boundary layer on shroud surface remained thicker than that on hub. Axial velocity from hub to shroud is evidence of continuing levelling in two boundary layer flow rates. Passage wake fairly distorted in radial velocity variations but a little wake still exists in S 8.

Secondary velocities were seen to be similar between S4 and S8, altering little in magnitude. The magnitude of radial velocity value increased particularly from shroud to hub sidewall. Radial velocity contours showed that flow was also meandering between pressure and suction sides, due to the effect of secondary flows at the exit of impeller section.

Turbulent Kinetic Energies

Turbulent kinetic energy is the major contributor to mixing losses encountered in diffuser. At S1 (Fig. 4), high level of turbulent kinetic energy (3 m^2/s^2) was observed in blade wake and passage wakes, with slightly lower values. This behaviour is in close agreement with reported studies[7,16]. High levels of turbulent kinetic energy occurred not only in regions of mixing, but also in regions of unsteady flow, where fluctuations were of low frequency and did not result in significant Reynolds stresses. At S2, levels of kinetic energy were significantly decreased in blade wake with largely mixed out. Kinetic energy levels in passage wake decreased a little from S1. Blade wake largely mixed out and moved to right from left hand side in case with S2. High level of kinetic energy was again observed due to meandering of passage wake across shroud surface. Even infrequent movement of wake towards suction side resulted in high levels of kinetic energy due to large difference in mean velocity between passage wake (3 m/s) and main flow (10 m/s).

Kinetic energy also showed a slight increase at S3, however high level of kinetic energy was still apparent at shroud due to meandering of passage wake. Peak in turbulent kinetic energy (comparing S4 to S8) was still located close to shroud wall. Relatively low kinetic energy levels were observed near hub wall rather than near shroud side, may be due to the movement of passage wake from pressure side to suction side on shroud wall.

Pressure inside impeller exhibited quite non-uniform especially around shroud side of impeller, but gradually moderated while reaching to diffuser section (Fig. 5). Circumferential variations were negligible at the last measurement station.

Conclusions

Diffuser flow at impeller exit was highly non-uniform, but developed into a Pouseille flow between diffuser walls. Strong axial cross velocities resulting from secondary flows within impeller delayed development

of axisymmetric at above design flow rate. Circumferential variations in velocity were prevented from mixing out rapidly at higher flow rate due to the presence of strong secondary flows. Blade wake mixed out very much quicker than passage wake because of the higher Reynolds stress levels. High turbulent kinetic energy within passage wake was due to low frequency meandering of wake position rather than high frequency turbulent mixing. Hub boundary layer was thinner than that on the shroud at last measurement station. Diffuser flow suggests that flow was almost completely mixed out at diffuser exit. Strong secondary flows inherited from the impeller due to pressure variations. Finally, it was impossible to avoid significant incidence losses with fixed vane diffuser, because of inherent dependence of flow angle on flow rate.

References

1 Stanitz J D, Some theoretical aerodynamic investigations of impeller in radial and mixing flow centrifugal compressors, Trans ASME, 74 (1952) 473-497.

2 Dean R. C Jr & Senoo Y, Rotating wakes in vaneless diffusers, ASME J Basic Engg, (1960), 563-570.

3 Johnston J R & Dean R C, Losses in vaneless diffuser on centrifugal compressor and pumps, J Engg Power, 88 (1966) 49-60.

4 Dean R C, On the unresolved fluid dynamics of the centrifugal compressor, Adv Centrifugal Compressors, ASME special publication, (1971) 1-55.

5 Senoo Y & Kinoshita Y, Influence of the inlet flow conditions and geometries of centrifugal vaneless diffusers on critical flow angle for reverse flow, ASME Trans, 99 (1977) 98-103.

6 Maksoud T M A & Johnson M W, Mean and turbulent flow measurements within the vaneless diffuser of a centrifugal compressor, Proc IMechE Conf, C263/87, 1987.

7 Maksoud, T M A & Johnson M W, Stress tensor measurements within the vaneless diffuser of a centrifugal compressor, Proc. IMech E Conf, C263/87, 1989.

8 Hathaway M D, Chriss R M, Wood J & Strasisar J, Experimental and computation of the NASA low-speed centrifugal compressor field, ASME Pap No. 92-GT-213, 1992.

9 Mounts J S & Brasz J J, Analysis of the jet/wake mixing in a vaneless diffuser, ASME Pap No. 92-GT-418, 1992.

10 Senoo Y & Nishi M, Prediction of flow seperation in a diffuser by a boundary layer calculation, J Fluids Engg, 99 (1977), 379-389.

11 Johnson MW & Moore J, The development of wake flow in a centrifugal compressor, ASME J Engg Power, 102 (1980) 383-390.

12 Pinarbasi A & Johnson M W, Detailed flow measurements in a centrifugal compressor vaneless diffuser, ASME J Turbomachinery, 116 (1994) 453-461.

13 Pinarbasi A & Johnson M W, Detailed stress tensor measurement in a centrifugal compressor vaneless diffuser, ASME Pap 94-GT-227, 1994.

14 Pinarbasi A & Johnson M W, Off design flow measurement in a centrifugal compressor vaneless diffuser, ASME J Turbomachinery, 117 (1994) 602-608.

15 Johnson M W & Moore J, Secondary flow and mixing losses in a centrifugal compressor impeller, ASME J Engg Power, 105 (1983) 24-32.

16 Pinarbasi A & Johnson, M. W, 3-D flow measurement in conical and straight wall centrifugal compressor vaneless diffuser, J Mech Engg Sci, 210 (1996) 51-61.