1998

Secondary Flow in Return Channels and Its Effect on Multistage Compressor Performance

J. J. Brasz
Carrier Corporation

SECONDARY FLOW IN RETURN CHANNELS AND ITS EFFECT ON MULTISTAGE COMPRESSOR PERFORMANCE

by

Joost J. Brasz
Carrier Corporation
Syracuse, NY 13221
joost.j.brasz@carrier.utc.com

ABSTRACT

Disappointing initial performance of a 6,000 cfm, 3:1 pressure ratio, high specific speed, two-stage compressor was traced down to a mismatch between the flow leaving the return channel and the second-stage impeller. A CFD flow field analysis showed the origination of a vortex in the deswirl vane section of the return channel. The original second-stage impeller design did not account for this vortex. It was designed, as is standard practice in multi-stage compressor design, for flow with constant inlet flow angle from hub to shroud. As a result of the vorticity created by the return channel, the existing second-stage impeller suffered incidence losses of opposite sign on hub and shroud. A new second-stage impeller was designed for minimum incidence losses from hub to shroud based on the calculated exit flow field of the return channel. Performance improvement was confirmed when the compressor was tested with this redesigned second-stage impeller.

INTRODUCTION

The geometry of the return channel is defined as all the gas passages between the exit of the diffuser of the previous stage and the inlet of the impeller of the next stage. As such it exists of (see Figure 1):

1. a crossover bend where the meridional flow component makes a 180 degree turn from radial outward to radial inward,
2. a set of deswirl vanes which remove the tangential component of the radial inward flowing fluid, changing the flow angle by 60 to 70 degrees,
3. a return channel exit bend where the flow is turned 90 degrees from radial inward to axially outward to enter the next stage impeller.

Previous studies on return channels were mainly experimental and concentrated on loss reduction of the channel passage by determining the optimum shape for each of these components. These studies investigated the optimum radius of curvature and diffusion ratio of the cross-over bend, the optimum inlet radius of the deswirl vanes, the aerodynamic loading of the deswirl vanes and the design of the return channel exit bend including the insertion of turning vanes [1-5].

During the last ten years, existing single-stage HCFC-22 centrifugal compressors have been redesigned for HFC-134a. This redesign consists of an increase in the through-flow areas of the critical aerodynamic gas passages inside the compressor, such as the impeller and the diffuser, to accommodate the 50% larger volumetric flow rates required by HFC-134a to maintain original HCFC-22 chiller capacity [6]. A similar approach to an existing two-stage HCFC-22 compressor with 2000 ton chiller capacity needed, besides the aerodynamic redesign of the first- and second-stage impellers and diffusers, a redesign of the return channel for 50% larger volumetric flow rate. Cross-sections of the existing and the newly designed two-stage 2000 ton compressor are shown in Figures 2 and 3. The high specific speed of the redesigned stage was beyond the existing

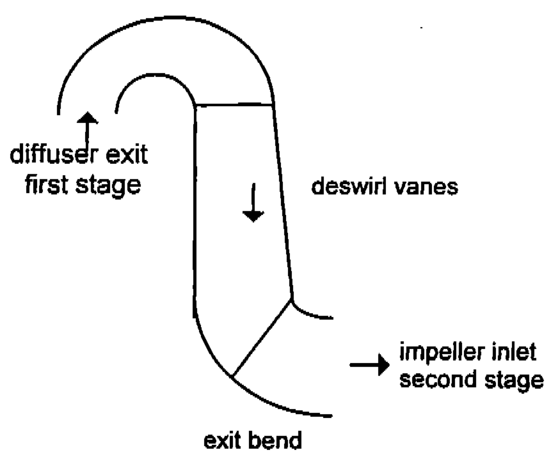

Figure 1. *Return channel geometry definition*

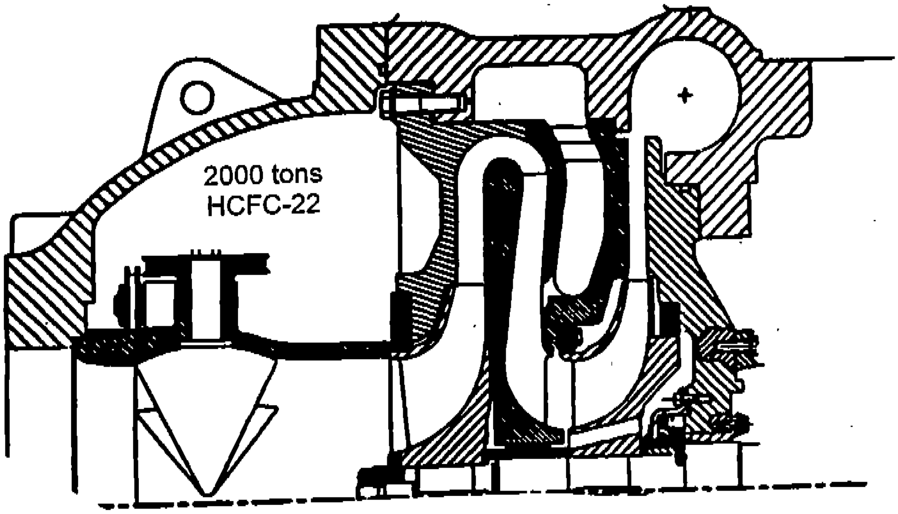

Figure 2. Cross-section of 2000 ton two-stage centrifugal compressor using HCFC-22

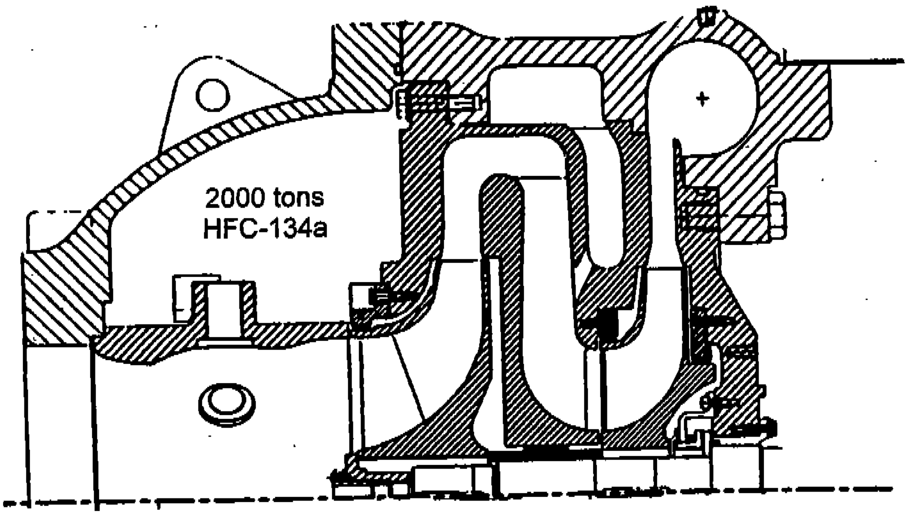

Figure 3. Cross-section of redesigned 2000 ton two-stage centrifugal compressor using HFC-134a

experience base for multistage compressors. However, performance calculations using an in-house mean-streamline code predicted acceptable efficiency levels for the redesigned two-stage compressor. When the performance of the initial two-stage HFC-134a retrofit compressor fell short on expectations, a computational fluid dynamics (CFD) analysis of the return channel was undertaken.

CFD ANALYSIS

The grid generation package CFD-GEOM was used to generate the computational grid from an IGES file containing the solid model of the return channel. The computational domain of the return channel was extended with the vaneless diffuser on the inlet side and an axial extension of the discharge plane of the return channel at the outlet side in order to impose constant boundary conditions. The geometry and the computational grid are shown in Figure 4. The details of the leading edge and trailing edge of the deswirl vanes and the fillets of the intersections of the blade walls and the hub and shroud have been modeled carefully by increasing the grid point density in those areas. A five-zone multi-domain grid was generated to facilitate the domain decomposition and gridding process. The coarse grid had 51,500 and the final grid had 126,500 grid cells, respectively. The flow field is periodic across the passage and thus only a single blade passage was simulated. The flow field was simulated using the general purpose code CFD-ACE.

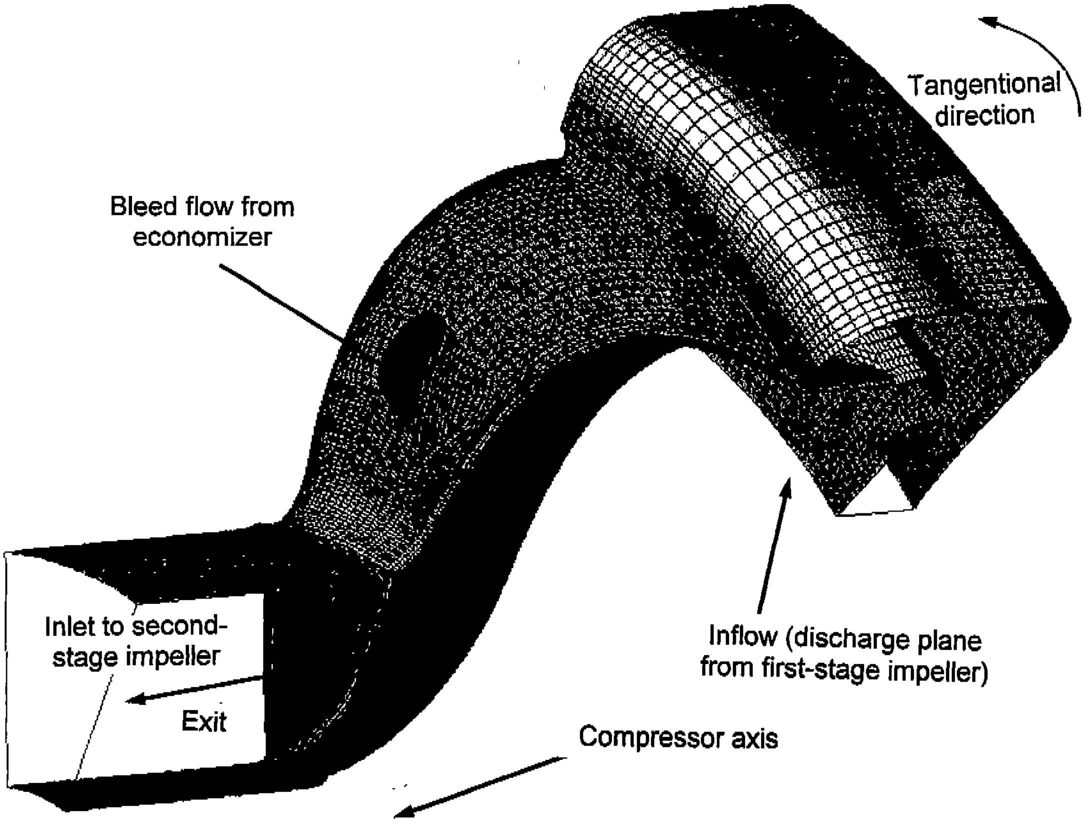

Figure 4. Computational grid for the return channel of the 2,000 ton HFC-134a compressor.

BOUNDARY CONDITIONS

The following boundary conditions were used for the solution:

1. Periodicity in the tangential direction.
2. No-slip and adiabatic conditions for all solid surfaces.
3. Inlet velocities specified as profiles of the distance from hub to shroud (obtained by circumferential averaging the solutions at the exit of the first-stage impeller, carried out in separate CFD calculation).
4. Adjustment of the static pressure at the downstream plane until the inlet total pressure and total temperature match the predicted exit conditions of the first stage impeller.
5. Economizer mass flow rate set at 10.5% of the total mass flow rate.

OPERATING CONDITIONS

The calculation was performed with the following operating conditions:

1. The working fluid is HFC-134a modeled as a perfect gas. The compressible flow calculations were done with a constant dynamic viscosity of 1.2155×10^{-5} kg/(m.s), molecular weight 121.1 and specific heat of 600.51 J/(kg.K).
2. k-ε turbulence model with standard wall functions.
3. Central differencing of convective fluxes for velocity variables with a damping factor of 0.30. Upwind scheme for k, ε and energy equation.

CALCULATION RESULTS

The small radius of curvature of cross-over bend relative to its width causes a region of recirculation on the convex side of the bend and a very non-uniform exit through-flow (meridional) velocity profile as shown in Figure 5. The tangential component of the flow velocity leaving the cross-over bend is fairly uniform as can be seen from the first plot in Figure 7. As a result, the large variation in meridional velocity is

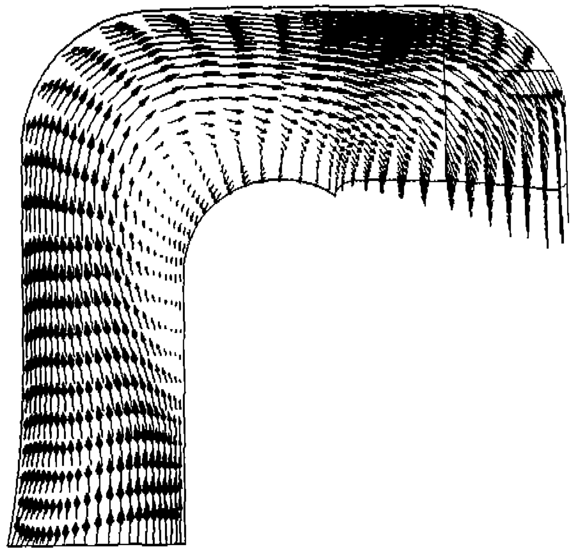

Figure 5. Meridional projection of velocities in the vaneless diffuser and the cross-over bend

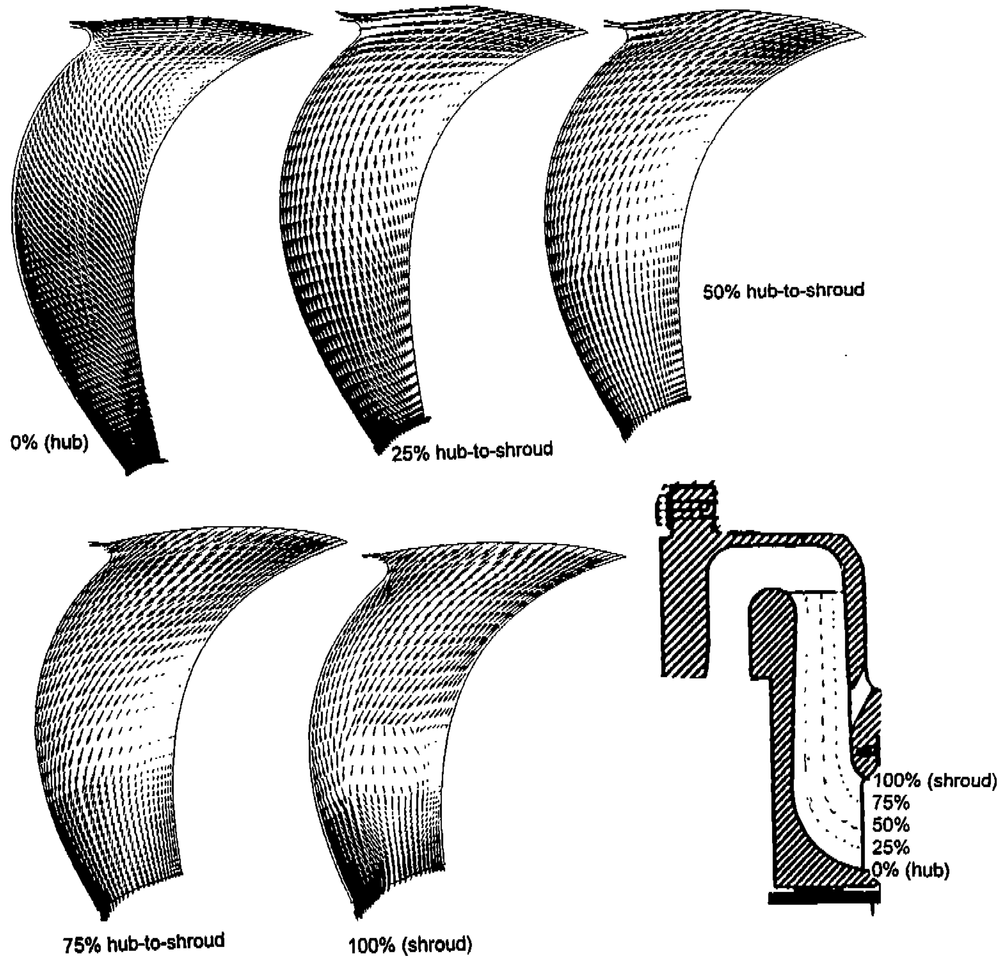

Figure 6. Velocity plots in the circumferential (blade-to-blade) plane at various hub-to-shroud locations

accompanied by a large variation in flow angle from hub to shroud, with the flow direction at the hub being very tangential (corresponding to a limited through-flow component) and at the shroud much more radial.
Figure 6 shows the velocities in the blade-to-blade plane of the deswirl vanes at different hub-to-shroud locations. Close to the hub the flow at the inlet (top) is mainly tangential flowing from suction side to pressure side. Further down in the deswirl vane channel when the flow has a throughflow component the flow velocity direction is more radial than the blade orientation resulting in flow movement from pressure side to suction side. Close to the shroud the trend is just opposite. A large initial throughflow component causes the flow direction to be matched with the blade angle or slightly more radial. Further down in the deswirl vane passage the flow moves from suction to pressure side. At each of the hub-to-shroud stations there is a localized area of flow separation on the suction side of the vaned passage. At the hub side this region happens at the inlet of the deswirl vane section. Closer to the shroud this separation zone is located further down in the deswirl vane section. The entry of the economizer flow on the shroud side is also visible in the CFD results shown in Figure 6.
Figure 7 shows velocity projections on six successive tangential planes normal to the gas passage in the deswirl vane section. Ideally, these plots should show a continuous reduction of the tangential velocities resulting at completely swirl-free flow at the exit of the return channel. Instead a strong vortex forms in the area where most of the tangential turning takes place (between stations 1 and 3). This vortex weakens somewhat in the final (almost straight-bladed) section of the deswirl vanes. However, the exit flow leaving the deswirl vanes at station 6 still has an appreciable vortex component. As a consequence, the flow at the shroud has a pre-rotation in the direction of rotation of the second stage impeller and the flow at the hub has counter-rotation opposite to the direction of rotation of the impeller. An impeller which has been designed for swirl-free inlet flow, (the common design practice for multistage impellers which was applied to the original HFC-134a impeller) will see a large variation in incidence and corresponding incidence losses on hub and shroud as a result of this vortex. Performance, efficiency as well as capacity, will be affected by the presence of this vortex.
A new second-stage impeller was designed with the impeller inlet blade angles matching the flow angles leaving the return channel as calculated with this CFD analysis. Testing of the two-stage HFC-134a compressor with this redesigned second-stage impeller has confirmed the predicted performance improvement.

CONCLUSIONS

The inherently sharp flow turning in the crossover bend of high-specific speed multi-stage machines causes the creation of a very strong secondary flow phenomena culminating in an exit vortex in the same direction as the rotational direction of the impeller. The effect of this return channel exit vortex on downstream impeller performance can - at least for high specific speed multistage compressors - not be neglected without suffering a loss in performance. The negative performance effect of this vortex can be prevented by redesigning the next impeller for the appropriate inlet flow conditions.
Since conventional quasi-3D design methods do not capture the creation of this vortex, CFD analysis should always be performed for return channel design to determine the inlet conditions for the succeeding impeller. For lower specific speed multistage compressors the distortion of the flow in the cross-over bend is less severe and as a result the secondary vortex leaving the return channel is much weaker and can probably be neglected, as has traditionally been done.

References

1. Simon, H. and Rothstein, E. , "On the Development of Return Passages of Multi-Stage Centrifugal Compressors", ASME Symposium on "Return Passages of Multi-Stage Turbomachinery", June 20-22, 1983, FED- Vol.3, pp. 1-11.
2. Inoue, Y. and Koizumi, T., "An experimental Study on Flow Patterns and Losses in Return Passages for Centrifugal Compressor", ASME Symposium on "Return Passages of Multi-Stage Turbomachinery", June 20-22, 1983, FED- Vol.3, pp. 13-21.
3. Furst, R., "Experimental Evaluation of Diffusing Crossover Systems for a High-Pressure Liquid Hydrogen Pump", ASME Symposium on "Return Passages of Multi-Stage Turbomachinery", June 20-22, 1983, FED- Vol.3, pp. 45-52.
4. Meng, S.Y. and Jackson, E.D., "The continuous Diffusion Crossover System Design", ASME Symposium on "Return Passages of Multi-Stage Turbomachinery", June 20-22, 1983, FED- Vol.3, pp. 53-60.
5. Balligand, H., Huang, X. and Brasz, J.J., "The Effect of Impeller Inlet Annular Turning Vanes on Multistage Centrifugal Compressor Performance", paper 96-GT-257 presented at the 41st ASME International Gas Turbine & Aeroengine Congress, June 10-13, 1996, Birmingham, England.
6. Brasz, J.J., "Centrifugal Compressor Behavior with Alternate Refrigerants", paper 96-WA/PID-2 presented at the 1996 ASME International Mechanical Engineering Congress & Exhibition, Atlanta, GA, November 17-22, 1996.

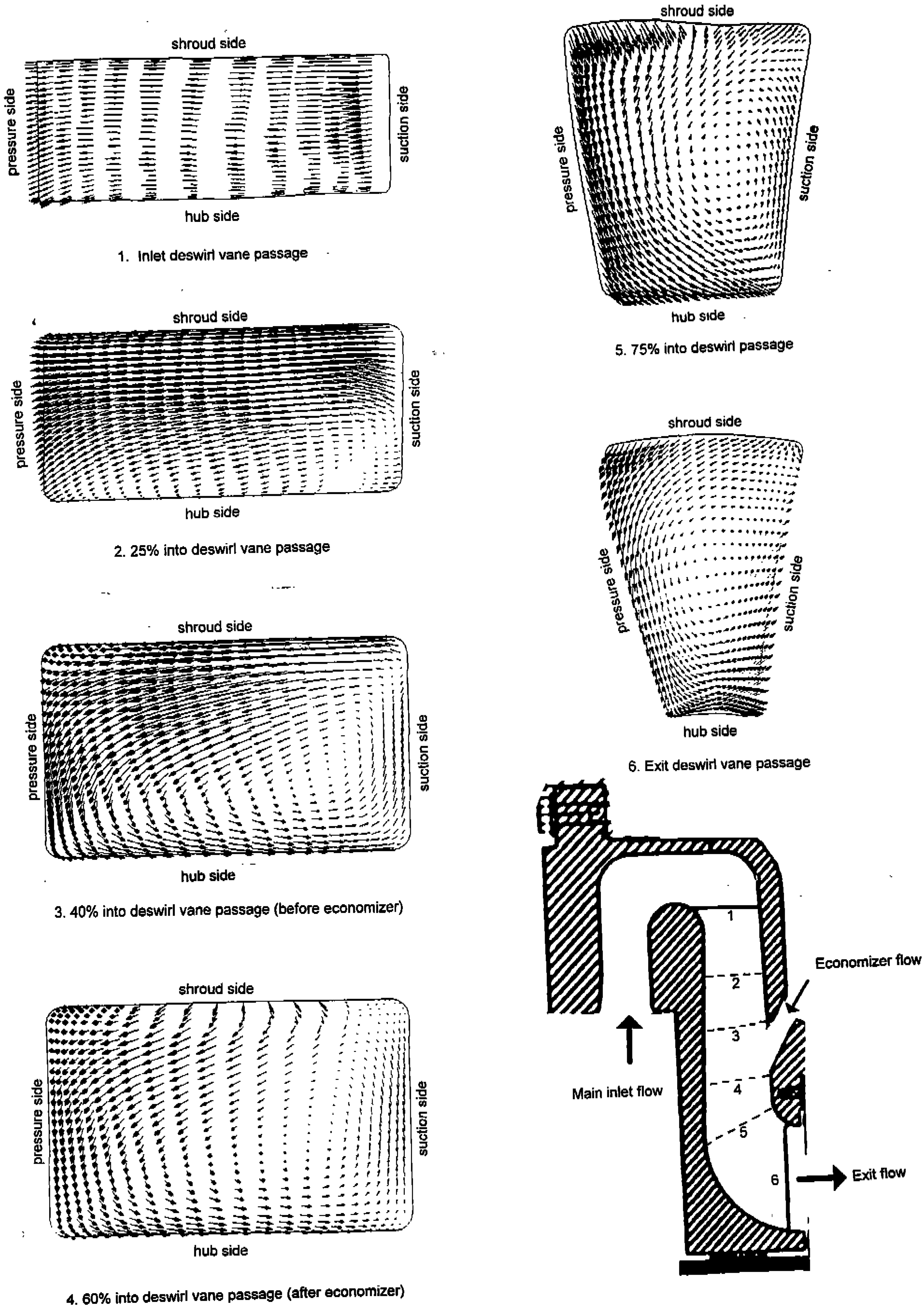

Figure 7. Velocity projections in six circumferential planes normal to the throughflow direction between inlet and exit of the deswirl vane section of the return channel.