Proceedings of the International Gas Turbine Congress 2003 Tokyo
November 2-7, 2003

IGTC 2003 Tokyo TS - 050

THEORY AND DESIGN OF THE REGENERATIVE FLOW COMPRESSOR

[1]Abraham Engeda and Mukarrum Raheel
Turbomachinery Lab
Michigan State University
Phone: 1-517-432 1834 E-Mail: engeda@egr.msu.edu
East Lansing, MI 48824

ABSTRACT

This paper presents a mathematical model to describe the complex three dimensional corkscrew flow pattern in a Regenerative Flow Compressor (RFC). All the major sources of losses in blade and channel region are identified. Governing equations for blade and channel region are developed. A 1-D performance prediction code for RFC based on governing equations and loss models is developed and performance results are compared with test data on Capstone multistage RFC used for natural gas compression in microturbine systems. Excellent agreement between theoretical and experimental results is observed, thus validating the proposed mathematical model. In order to make the discussion in this paper useful for designers and engineers, some design criteria are established through available experimental data in literature, test data on Capstone single and multistage RFC and results from design sensitivity analysis from the performance prediction code. The design procedure suggested at the end of this paper can be very useful for designing radial blade impeller and channel of RFC. Proposed design guidelines can serve to produce a preliminary design of RFC, which must be validated by performing CFD analysis later on. Currently work is in progress to perform CFD analysis on a RFC, which will be discussed in another publication to appear shortly.

NOMENCLATURE

A_b	Area of blade
A_c	Channel area
C_a	Axial clearance
C_r	Radial clearance
dX_G	Peripheral distance
MOT	Impeller tip Mach number
$\dot{m}$	Through mass flow rate
p	Pressure
P	Power
Q	Through volume flow rate
Q_c	Circulatory flow rate
R	Gas constant
$r_2, r_\circ$	Impeller tip and hub radius
r_1	Radius where streamline enters the blade
r_G	Centroidal radius
T	Torque
T_{in}	Absolute temperature of fluid at inlet port
U_1, U_2	Tangential velocity of impeller at hub and tip
V_c	Circulatory velocity
V_θ	Peripheral velocity
α	Shock loss parameter
β_2	Fluid angle at blade exit
σ	Slip factor
θ_p, θ_s	Pumping and stripper angle respectively
η_{iso}	Isothermal efficiency
ρ_{in}	Inlet fluid density
Φ	Specific mass flow rate
γ	Ratio of specific heats
ω	Rotational speed (RPM)

1. INTRODUCTION

A detail discussion on fundamentals and hypothesis of operation of regenerative flow compressors (RFC) can be found in Raheel and Engeda [1]. Main feature of RFC is their ability to generate high heads at low flow rates. They have a very low specific speed and share some of the characteristics of positive displacement machines such as a roots blower, but without problems of lubrication and wear. Multiple passes through the impeller blades (regenerative flow pattern) allows RFC to produce discharge heads of up to 15 times of those produced by centrifugal compressor operating at same tip speed. Cross sectional area of the peripheral flow in RFC is usually smaller than cross sectional area of the radial flow in a centrifugal compressor, which makes RFCs to operate at flows which are lower than the flows of a centrifugal compressor having an equal diameter and operating at an equal tip speed. These high-head, low-flow characteristics

Manuscript Received on March 18, 2003

of RFC make it well suited to a number of applications where a reciprocating compressor, a rotary displacement compressor or a low specific speed centrifugal compressor would not be as well suited. When a regenerative flow turbomachine is applied as a gas compressor, there is a further advantage of no surge or stall instability. Compression process in a regenerative compressor is usually not regarded as efficient. Typically regenerative compressors have an efficiency of less than 50% but still they have found many applications because they allow the use of fluid dynamic compressors in place of positive displacement compressors for duties requiring high head and low flow rates. Relative simplicity of construction and stable operating characteristics of RFC are making them more and more attractive to compressor users in several areas, including chemical, petroleum, and nuclear industries. Because of the reliability, compact size and low maintenance, recently there is an increasing use of regenerative compressors in low-pressure (0.2-15 psig) natural gas compression required by microturbine systems. Application of RFC in microturbine systems is discussed in Raheel and Engeda et al [2].

Typical cross section of a RFC is shown in Figure (1).

Flow through a regenerative compressor resembles a corkscrew pattern as illustrated in Figure (1) with the help of a streamline passing though the impeller. Figure (2) is an enlarged view of section (A-A) in Figure (1) and it shows flow around the blade of a regenerative compressor. In contrast to other popular types of continuous flow compressors in which the fluid passes through the impeller once, the regenerative compressors have the fluid exposed to the impeller many times. The repetition of the action of the impeller blading on the fluid is in effect, “multistaging” which makes regenerative compressors capable of developing high pressure ratios in a single stage. There is additional energy imparted to the fluid each time it passes through the blades of the impeller, allowing substantially more motive force to be added which enables much higher pressures to be achieved in a more compact compressor design. Each passage through the vanes may be regarded as a conventional stage of compression. This regenerative flow pattern of regenerative turbomachines was first explained in Wilson et al [3]. Essential elements of a typical RFC are shown in Figure (1).

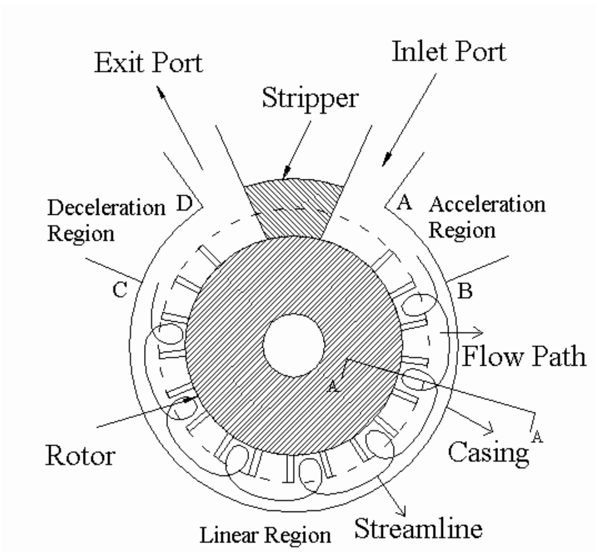

Figure (1) A Regenerative Compressor

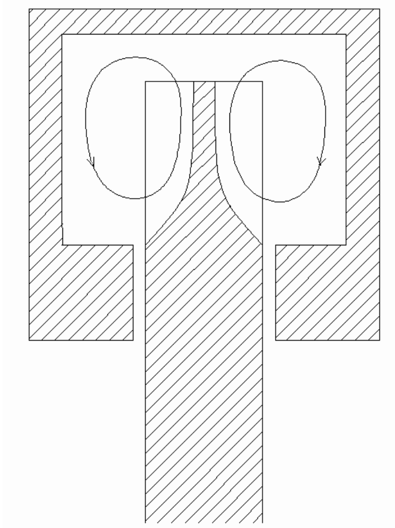

Figure (2) Section A-A Enlarged (Radial Blade)

Typically, a regenerative compressor has an impeller, inlet port, discharge port, stripper, flow passage and a casing. Regenerative compressors utilize a free rotating impeller just like other types of turbocompressors. Impeller has blades machined into each side at its periphery, which produce a series of helical flows, returning the fluid repeatedly through the blades for additional energy as it passes through an open annular channel. Inlet and discharge ports connect the external system piping to the flow channel. Fluid enters the flow channel via the inlet port, which is shaped to set up spiral flow around the annular channel. Fluid at high pressure is discharged to the piping from the discharge port. Between discharge and inlet, the casing clearance is reduced to block the high-pressure discharge from the low-pressure inlet. At this region, which is also known as stripper, the open channel closes to within a few thousandths of an inch of the sides and tip of the rotor and allows only the fluid within the impeller to pass through the suction. Clearances between the impeller disk and casing are kept to a minimum to prevent leakage from high-pressure side of the compressor back to low-pressure side. Stripper forces the fluid to go out through discharge port. Stripper also helps to cause the establishment and maintenance of the regenerative flow pattern. Around the greater portion of the periphery, impeller blades project into an annular channel built in the casing. Flow channel has cross sectional area greater than that of the impeller vanes. It is this annular flow channel from which fluid circulates repeatedly through the impeller blades. The fluid between blades is thrown out and across the annular channel.

Pressure variation of the fluid as it circulates through a regenerative turbomachine for several flow rates is shown in Figure (3). These curves suggest five regions in the pump operation, which are also marked in Figure (1).

- Inlet region (A): The flow experiences some pressure loss through the inlet region.
- Acceleration region (A-B): The flow enters the working section of pump with a velocity and pressure dependent largely on the inlet region. Until the flow reaches fully developed pattern of linear region, the circulatory velocity changes.
- Linear region (B-C): The pressure gradient is constant as indicated in the diagram. This region is referred to as the working section of the pump where the flow pattern is fully developed.
- Deceleration region (C-D): In this region, a deceleration occurs and the kinetic energy of the circulatory velocity is changed as a pressure rise. Therefore, there is a little pressure rise as shown in Figure (5).
- Outlet region (D): A loss similar to that at the inlet region occurs at the outlet region.

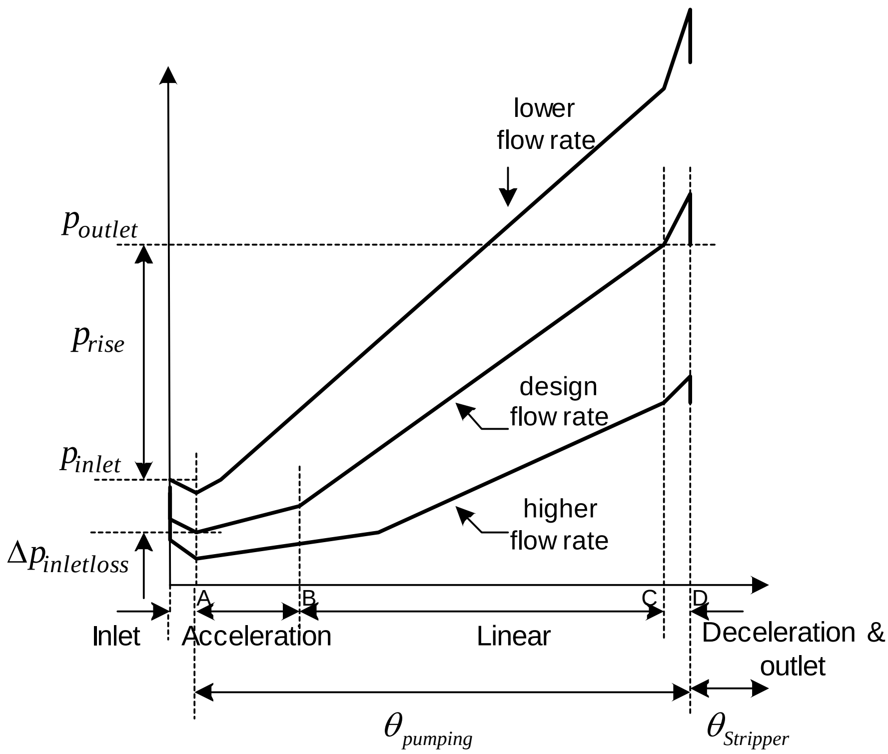

Figure (3) Tangential pressure variation in a regenerative turbomachine

There are very few mathematical models in literature which explain the behavior of regenerative turbomachines and predict the performance. Most of these models need extensive experimental support for performance prediction. Hence, it is very interesting from an industrial stand point to find efficient theoretical means which are able to forecast the regenerative turbomachine performances using easy to find geometric and fluid dynamic parameters.

Wilson et al [3] presented a simplified model to permit the development of a theoretical analysis of the three-dimensional fluid motion inside a regenerative pump. Basic model developed by Wilson was to represent the phenomenon in the linear region as shown in Figure (3). He made several assumptions and applied fluid dynamic equations to arbitrary control volumes of the pump. The entire pump flow was characterized by the tangential velocities V_t and circulatory velocity V_c along a mean streamline. Dimensionless performance characteristics for STA-RITE TH 7 regenerative pump were tested and reported in Wilson et al [3]. Experimental plots of flow versus head were obtained at seven different speeds with air as working fluid. Calculated and experimental performance curves were compared and excellent agreement was observed. Qualitatively, the relations presented by Wilson satisfied the major details of the observed flow phenomena, but they needed extensive experimental support to predict performance. Moreover, his model assumed incompressible flow, thus unable to predict performance of compressors producing high pressure ratio. Moreover, Wilson model lacked in correlation of losses with geometric and aerodynamic parameters. In this paper, authors have extended Wilson's work to incorporate compressible flow to predict the performance characteristic of multistage RFC employed by Capstone Turbine Corporation for compression of natural gas for Model 30 microturbine. Moreover, authors have done some 1-D loss modeling to correlate various losses with geometric and aerodynamic parameters, thus eliminating need of experimental support. Details of the loss modeling can be found in Raheel [4].

2. MATHEMATICAL MODEL

Mathematical formulation is based on an arbitrary element of depth $dX_G = r_G d\theta$ in the peripheral direction of the compressor as shown in Figure (4). Equations of motion can be derived by considering these arbitrary small elements of one side blade and channel. Dimensions of the impeller and flow channel are given symbolically in Figure (5) where points 1 and 2 denote the locations at which assumed streamline enters and leaves the impeller respectively. For simplicity in the presentation of the analysis, wall friction and other irreversibility are introduced as head losses.

2.1 Assumptions

Following assumptions are made in the model.

- Fluid is assumed incompressible within a control volume, however from one control volume to the next, fluid density changes.
- Fluid shear is assumed negligible in the model, however it is considered later in the calculation by assuming a model for tangential head loss.
- Steady flow without any leakage. Leakages are considered later based on a model, however to avoid complexity in mathematics, leakages are assumed zero.
- Characteristic flow is one-dimensional in which major direction is (radial, tangential, and axial). The actual path traversed by a streamline is shown in Figure (5).
- There are no end effects of suction, discharge and stripper carryover. The inlet and exit loss are considered in a separate model.
- Tangential pressure gradient is independent of radius. Literature and experimental studies [3] have confirmed that this assumption is quite valid.
- Although, tangential pressure gradient around the periphery is not perfectly linear, however for simplicity, assumption of linear pressure rise across the periphery is reasonable.

2.2 GOVERNING EQUATIONS

The tangential components of velocity at points 1 and 2 can be given as

$$V_{\theta 1} = \alpha U_1 \tag{1}$$

$$V_{\theta 2} = \sigma U_2 \tag{2}$$

where σ is the slip factor and α is a shock loss parameter introduced at blade entrance to quantify shock losses. More details on calculation of slip factor and shock loss parameter can be found in Raheel [4].

Circulatory velocity at point 1 and 2 is assumed to be equal. Thus,

$$V_{c1} = V_{c2} = V_c \tag{3}$$

Circulatory flow rate can be approximated by

$$dQ_c = C_r V_c dX_G \tag{4}$$

Applying angular momentum equation to the impeller control volume of Figure (4), we get

$$dT = \rho dQ_c \left(r_2 \sigma U_2 - r_1 \alpha U_1 \right) + r_G A_b \frac{dp}{d\theta} d\theta \tag{5}$$

Last term on the right hand side in the above equation represents the work done in raising the pressure of the fluid between the impeller blades. This pressure rise is utilized across the stripper to perform a turbine work. Therefore this term is ignored generally. Power input to the differential control volume can be given as

$$dP = \omega dT \tag{6}$$

Thus we can write,

$$dP = \rho dQ_c \left(\sigma U_2^2 - \alpha U_1^2 \right) \tag{7}$$

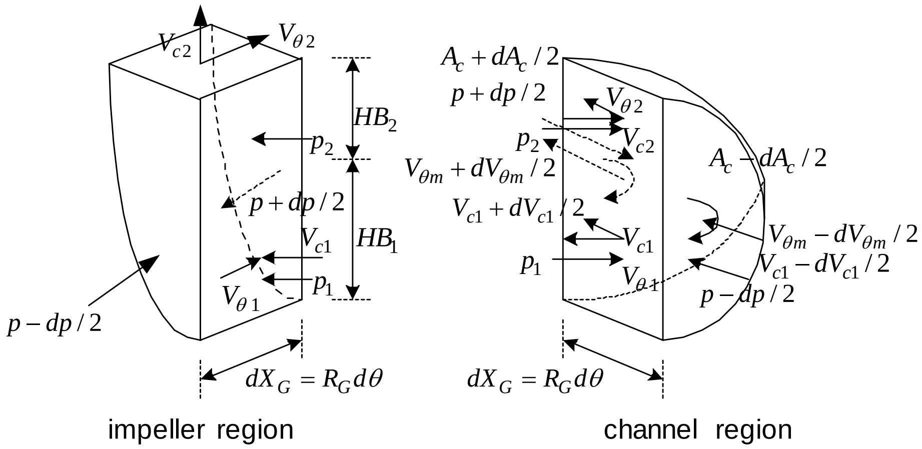

Figure (4). Control volumes representing section $d\theta$ of open channel and impeller

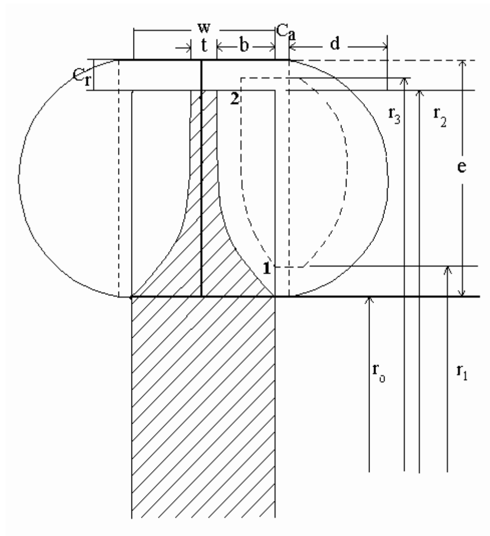

Figure (5). Schematic of blade and channel geometry

Applying Bernoulli's equation to the impeller control volume of Figure (4), we get

$$dP = \rho dQ_c \left(\frac{\sigma^2 U_2^2}{2} + \frac{V_{c2}^2}{2} + \frac{p_2}{\rho} - \frac{\alpha^2 U_1^2}{2} - \frac{V_{c1}^2}{2} - \frac{p_1}{\rho}\right) \tag{8}$$

Equations (7) and (8) yield,

$$\frac{p_1}{\rho} + \frac{V_{c1}^2}{2} + \frac{\alpha^2 U_1^2}{2} + \sigma U_2^2 = \frac{p_2}{\rho} + \frac{V_{c2}^2}{2} + \frac{\sigma^2 U_2^2}{2} + \alpha U_1^2 + gH_{cb} \tag{9}$$

where gH_{cb} is the head loss of circulatory velocity through the impeller region.

Applying continuity equation to the channel control volume of Figure (4),

$$\rho(V_{\theta m} + dV_{\theta m}/2)(A_c + dA_c/2) + \cancel{\rho dQ_{c1}} = \rho(V_{\theta m} - dV_{\theta m}/2)(A_c - dA_c/2) + \cancel{\rho dQ_{c2}} \tag{10}$$

or

$$d(V_{\theta m} A_c) = 0 \tag{11}$$

where, $V_{\theta m}$ is mean tangential velocity which can be obtained as

$$V_{\theta m} = \frac{Q}{A_c} \tag{12}$$

Angular momentum equation in the tangential direction yields

$$\rho R_G (V_{\theta m} + dV_{\theta m} / 2)(A_c + dA_c / 2) dV_{\theta m} + \rho dQ_c (R_1 V_{\theta 1} - R_2 V_{\theta 2})$$
$$= R_G (P - dP/2)(A_c - dA_c / 2) - R_G (P + dP/2)(A_c + dA_c / 2)$$
$$+ R_G P dA_c - wall\ friction\ term \qquad (13)$$

By use of continuity equation, the above equation can be reduced to the following equation.

$$dgH = dQ_c / Q_s (U_2 V_{\theta 2} - U_1 V_{\theta 1}) + V_{\theta m}^2 dA_c / A_c - dgH_L \qquad (14)$$

where $Q_s = \omega R_G A_c$ is flow rate based on solid body rotation.

In the right side of the above equation, the first term refers to head rise caused by momentum exchange of blade, the second term gives head rise caused by the deceleration of the mean tangential velocity and the last term gives head loss caused by the friction and the contraction or expansion of the tangential velocity.

Applying Momentum equation in the circulatory direction

$$\cancel{\rho dQ_{c1} V_{c1}} - \cancel{\rho dQ_{c2} V_{c2}} + \rho Q dV_c = p_2 H B_2 dX_2$$
$$- p_1 H B_1 dX_1 - wall\ friction\ term \qquad (15)$$

$$\frac{Q V_c dV_c}{dQ_c} = \frac{p_2 - p_1}{\rho} - gH_{cc} \qquad (16)$$

where gH_{cc} is the head loss of circulatory velocity through the impeller region.

Energy equation applied to channel control volume of Figure (4) after some simplifications yields

$$\frac{p_2}{\rho} + \frac{V_{c2}^2}{2} + \frac{\sigma^2 U_2^2}{2} = \frac{p_1}{\rho} + \frac{V_{c1}^2}{2} + \frac{\alpha^2 U_1^2}{2}$$
$$+ gH_{cc} + \frac{Q}{dQ_c}(dgH + dgH_L - V_{\theta m}^2 dA_c / A_c + V_c dV_c) \qquad (17)$$

If equation (9), equation (14) and equation (16) are used in equation (17), the first order nonlinear ordinary differential equation can be obtained for the circulatory flow as follow.

$$\frac{Q V_c dV_c}{dQ_c} = (1 - \frac{Q}{Q_s})(\sigma V_{\theta 2}^2 - \alpha V_{\theta 1}^2) - gH_c \qquad (18)$$

where $gH_c = gH_{cb} + gH_{cc}$ is the sum of head loss related to the circulatory velocity.

3. Slip and shock losses

Tangential pressure gradient in regenerative turbomachines enhances the slip factor considerably. Pressure difference between any two adjacent blades of an impeller causes a tendency for a secondary circulation about each blade such that the fluid leaving the impeller deviates from the path prescribed by the blade surface, backwards with respect to the positive direction of impeller rotation. The result is that the fluid tangential velocity at exit is less than that which would be expected from the velocity triangle based on the outlet blade angle. In order to allow for the reduction in the ideal tangential velocity, a slip factor is usually introduced. It is defined as ratio between actual tangential velocity and the one obtained with the assumption that the flow angle and blade angle are identical. In present investigation, a model for the calculation of slip factor for regenerative turbomachines is introduced, details of which can be found in Raheel [4]. Shock or incidence losses are caused by difference between blade angle and flow angle when fluid enters the blades. Wilson [3] introduced a shock loss parameter α to quantify such losses. Wilson [3] model is used in this work to calculate shock losses.

4. Circulatory Head Losses

Circulatory head losses have two contributions.

- Head loss of circulatory velocity through the impeller region is referred as gH_{cb}.
- Head loss of circulatory velocity through the channel region is referred as gH_{cc}.

The sum of these two head losses is the total circulatory head loss given as:

$$gH_c = gH_{cb} + gH_{cc} \qquad (19)$$

Circulatory head losses arise from many sources. Following sources of circulatory head loss were quantified. Details can be found in Raheel [4].

Channel turning losses (k_t) are due to 180^o turn of the fluid through the channel.

Blade turning losses (k_b) are due to the turning of fluid through the impeller blades

Channel and blade mixing losses (k_{ch}, k_s) occur due to the mixing of fluid leaving the tip of the impeller blade and mixing with incoming stream of flow through the channel

Sudden expansion losses (k_{se}) are caused by the sudden increase in flow area when fluid flows from blades to channel.

Consequently, model for the circulatory head loss can be arranged as follows.

$$gH_c = \frac{1}{2}k_t V_c^2 + \frac{1}{2}k_b V_c^2 + \frac{1}{2}k_{ch} V_c^2 + \frac{1}{2}k_s V_{\theta m}^2 + \frac{1}{2}k_{se} V_c^2 \qquad (20)$$

The various loss coefficients in above equation are correlated with geometric and aerodynamic parameters in Raheel [4].

5. Tangential Head Losses

Head loss caused by channel friction is referred as tangential head loss denoted by dgH_L. It involves the channel curvature effect and it can be determined by applying the classic pipe-loss formula.

$$dgH_L = \frac{\lambda_f V_{\theta m}^2 dX_G}{2D_h} \qquad (21)$$

where

$$\lambda_f = \lambda_o \left(1 + 0.075\,\mathrm{Re}^{0.25}\left(\frac{D_h}{2R_{tip}}\right)^{0.5}\right) \qquad (22)$$

λ_o is defined for straight channel as,

$$\lambda_o = 0.316\,\mathrm{Re}^{-0.25} \qquad (23)$$

where Re is given based on Hydraulic diameter as,

$$\mathrm{Re} = \frac{D_h V_{\theta m}}{\nu} \qquad (24)$$

6. Leakage Losses

Total leakage flow rate can be estimated by the following equation suggested by El-Hag [5].

$$Q_{leak} = \frac{\omega r_2}{2}\left(C_r b + \frac{C_a r_2}{2}\right) + 2C_D \omega r_2 \sqrt{\frac{2\frac{d\Psi}{d\theta}}{Z_s}}\left(C_r\left(b + C_a\right) + C_a\left(r_2 - r_o\right)\right) \qquad (25)$$

7. Losses in Ports

Losses in inlet and discharge ports are estimated by

Inlet port

$$\Delta P_{in} = \frac{1}{2}k_{in}\rho V_{in}^2 \qquad (26)$$

Outlet port

$$\Delta P_{out} = \frac{1}{2}k_{out}\rho V_{out}^2 \qquad (27)$$

Based on proposed mathematical model, a performance prediction code was developed to predict Capstone four stage RFC performance. The code takes geometry data, gas properties and operating point information as input and predicts the pressure rise for the entire operating range. Predicted results were then compared with test data for Capstone multistage RFC presented by Raheel and Engeda et al [2]. Excellent agreement was observed as seen from Figure (11) and (12) which represent theoretical and test data overlapped for natural gas as working fluid at four inlet pressures of $0\ psig$, $5\ psig$, $10\ psig$ and $15\ psig$ at inlet temperature of $70^\circ F$. To condense the presentation, Figure (11) represents results at $195\ slpm$ and Figure (12) represents results at $250\ slpm$ flow rate.

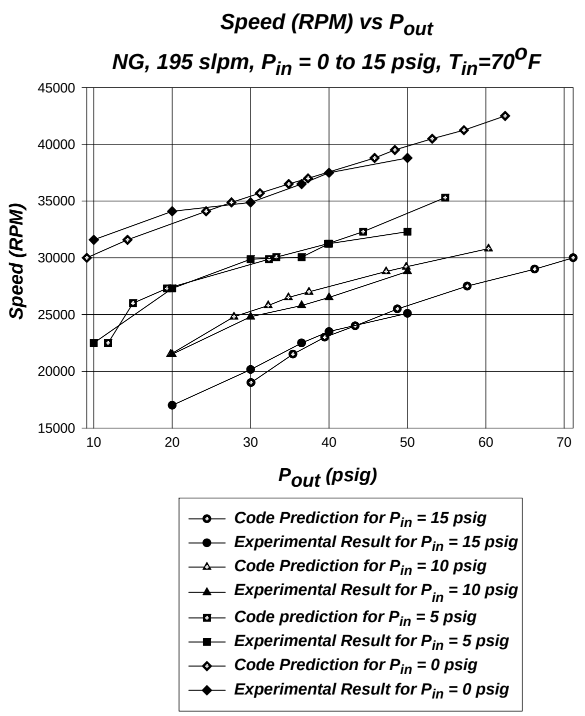

Figure (11) Theoretical and test data overlapped for flow rate $195\ slpm$

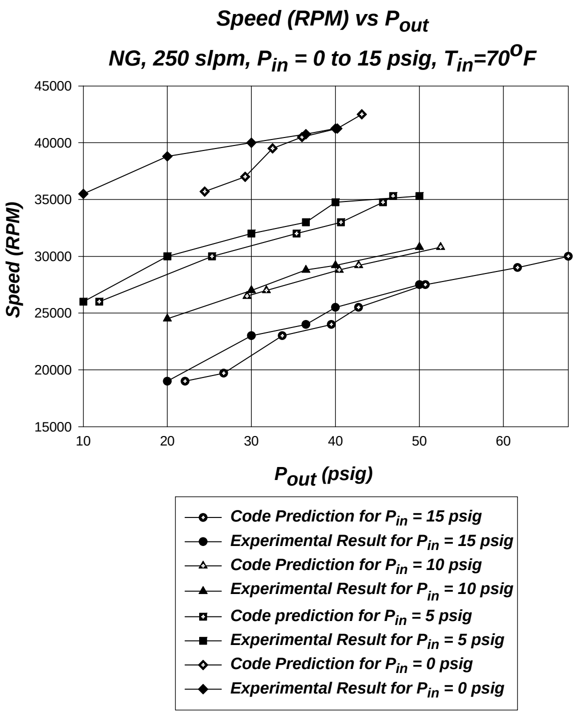

Figure (12) Theoretical and test data overlapped for flow rate $250\ slpm$

8. Design sensitivity analysis

After predicting the performance, it is desired to vary some of the geometric parameters in order to study which design changes can help to improve RFC performance. Usually, it is difficult to optimize the design for all operating points. Therefore, it is a standard practice in turbomachinery to select a design point where designer intends to achieve peak efficiency. Defining two non-dimensional parameters; impeller tip Mach number and specific mass flow rate as,

Impeller tip Mach number $$M_{OT} = \frac{r_2\omega}{\sqrt{\gamma R T_{in}}} \tag{28}$$

Specific mass flow rate $$\Phi = \frac{\dot{m}}{4r_2^2 P_{in}}\sqrt{\frac{RT_{in}}{\gamma}} \tag{29}$$

For Capstone microturbine operation, it is reasonable to consider following design point where multistage RFC has to operate,

$$P_{in} = 0\ psig, T_{in} = 70^{\circ} F\ \ M_{OT} = 0.369, \Phi = 1.04\times 10^{-3}$$

Extensive sensitivity analysis is carried out using the code by varying some of the channel and blade dimensions listed below and trying to maximize head at the design point. These dimensions are also shown in Figure (5).

1. Radial clearance "C_r"
2. Channel inlet area (by varying channel depth at station A denoted by "d_A")
3. Channel height "e"
4. Area ratio of inlet and outlet channel $A_{c,A}/A_{c,B}$
5. Impeller hub radius $r_{\circ}$
6. Impeller tip radius r_2
7. Number of impeller blades "Z"

It must be noted that station A is defined close to inlet of the compressor and station B is defined close to the discharge port. The details of sensitivity analysis and suggested design changes for performance improvement of Capstone multistage RFC can be found in Raheel [4]. The purpose of this research paper is not only investigating the flow mechanism inside a regenerative turbomachine, but also to make the discussion useful for engineers and designers. Hence it is essential from an industrial stand point to establish some design criteria and guidelines for impeller and channel design of RFC, which can serve to produce a preliminary design.

9. Design Guidelines

There is no evidence available in literature about the existence of any design criteria and guidelines for regenerative compressor design. Thus need was felt to look into test data published in literature, test data on Capstone single and multistage RFC presented in Raheel and Engeda et al [2] and sensitivity analysis results from the performance prediction code to establish useful design criteria which can serve as a starting point in regenerative compressor design. Moreover, authors propose a design procedure to be followed for sizing dimensions of impeller and channel of radial blade RFC. Details of the methodology of establishing the design criteria and design procedure can be found in Raheel [4]. Following design criteria and guidelines are proposed after this comprehensive analysis.

To avoid the compressibility effects, it is suggested to choose impeller tip Mach number to be less than 0.8, thus

$$M_{OT} < 0.8 \tag{30}$$

A design criterion was established for sizing the radial clearance and channel depth given as,

$$0.2 < C_r / d_A < 0.65 \quad (31)$$

$$0.03 < C_r / r_2 < 0.05 \quad (32)$$

The channel area at station A can be found by

$$A_{c,A} = \frac{\pi d_A^2}{2} \quad (33)$$

The impeller hub radius can be calculated using

$$r_{\circ} = r_2 + c_r - 2d_A \quad (34)$$

A design criteria for the area ratio at station A and station B was established

$$1.15 < A_{c,A} / A_{c,B} < 1.35 \quad (35)$$

Moreover, it is suggested that the number of impeller blades must be selected using the following design criteria

$$75 < Z < 90 \quad (36)$$

For best performance, the blade chevron angle must be selected using the following design criteria

$$45^{\circ} < \beta < 60^{\circ} \quad (37)$$

10. CONCLUSIONS

Theory and design of regenrative flow compressors is presented in this paper. A Performance prediction code is developed based on proposed mathematical formulation. Excellent agreement is observed in comparison of theoretical and experimental results for Capstone multistage RFC. Some design criteria and guidelines for impeller and channel design of radial blade RFC are presented for the first time in this paper. The proposed design methodology can be very useful from an industrial stand point for the preliminary design of radial blade RFC. The initial design resulting from this procedure has to be validated using CFD. Currently, authors are working on CFD analysis of a RFC, details of which will be reported in a publication to appear shortly.

ACKNOWLEDGMENT

Authors are thankful to Douglas Hamrin, Greg Rouse, Greg Priddie and Murali Chinta at Capstone Turbine Corporation, Chatsworth, CA for providing information related to geometry and raw test data to compare and validate theoretical results.

REFERENCES

1. Raheel, M., Engeda, A., (Michigan State University) “Current Status, Design and Performance Trends for the Regenerative Flow Compressors and Pumps”, (IMECE 2002-39594).

2. Raheel, M., Engeda, A., et al, “An Investigation of Performance Characteristics of Single-stage and Multi-stage Regenerative Flow Compressors (RFC)” (to appear).

3. Wilson W.A., Santalo M.A., Oelrich J.A., “A Theory of the Fluid-dynamic Mechanism of Regenerative Pumps”, Trans. of ASME, Nov. 1955, pp 1303 1316.

4. Raheel, M., “A Theoretical, Experimental and CFD analysis of regenerative flow compressors and regenerative flow pumps for microturbine and automotive fuel applications”, Ph.D. thesis, Michigan State University, USA, 2003.

5. El Hag, A.I. “A theoretical analysis of the flow in regenerative pumps”, Ph.D. thesis, University of Bath, England, 1979.

6. Cates P.S., “Peripheral-Compressor Performance on Gases with Molecular Weights of 4 to 400”, A.S.M.E. Transactions 64-WA/FE-25, Meeting November 29-December 4, 1964.

7. Burton D. W., “Review of Regenerative Compressor Theory”, Rotating Machinery for Gas-Cooled Reactor Application, TID-7631, April 2-4,1962 pp. 228-242.

8. Iversen H.W., “Performance of the Periphery Pump”, Trans. of ASME, Jan. 1955, pp 19 28.

9. Mason S. C., “Influence of Internal Geometry upon Regenerative Pump Performance”, M.I.T. B.S. thesis, 1957.

10. Oelrich J. A., “Development of an Analysis of a Regenerative Pump”, M. I. T. Ph. D Thesis, 1953.