Design Principles and Measured Performance of Multistage Radial Flow Microturbomachinery at Low Reynolds Numbers

This paper introduces and experimentally demonstrates the design concept of multistage microturbomachinery, which is fabricated using silicon microfabrication technology. The design process for multistage microscale turbomachinery based on meanline analysis is presented, along with computational fluid dynamics predictions of the key aerodynamic performance parameters required in this design process. This modeling was compared with a microturbine device with a 4 mm diameter rotor and 100 μm chord blades, based on microelectromechanical system technology, which was spun to 330,000 rpm and produced 0.38 W of mechanical power. Modeling suggests a turbine adiabatic efficiency of 35% and Re≈266 at the maximum speed. The pressure distribution across the blade rows was measured and showed close agreement with the simulation results. Using the model, the microturbine is predicted to produce 3.2 W with an adiabatic efficiency of 63% at a rotor speed of 1.1×10^6 rpm. [DOI: 10.1115/1.2979010]

1 Introduction

Recently, silicon turbomachines have been developed for various applications, such as power generation, propulsion, compression, and pumping [1,2]. Although there are limitations in the choice of structural shapes due to the 2D characteristic of the microfabrication process, the potential benefits of high power per unit volume and low production cost due to batch processing are attractive [3,4]. To date, the development efforts of microscale turbomachines have mostly focused on those with just one pair of stator and rotor blade rows. However, a single stage configuration can only provide a limited pressure ratio, consequently restricting the thermodynamic efficiency of miniature thermodynamic machines. Thus the need for multiple blade rows has arisen from theoretical and practical aspects during the development of gas turbines [1] and steam turbines [5]. High pressure ratios are especially beneficial to achieve a reasonable efficiency in a Rankine cycle, which produces power by expanding high pressure superheated steam through a turbine. The working fluid is pressurized in liquid state by a pump, allowing high pressure ratios, which compensate for the typically lower operating temperatures than in gas phase Brayton power cycles.

In this context, the concept of the multistage turbine was suggested for the micro-Rankine cycle power generation [5]. The whole system consists of microfabricated heat exchangers, a pump, a generator, bearings, and the turbine. The system was designed to generate a few watts of electrical power from a source of heat with an overall energy conversion efficiency between 1% and 12% depending on the thermal conditions. As a core power conversion component, the efficiency and power output of the turbine proved to be critical for the performance of the whole system.

This paper will present the design principles of multistage microturbomachinery, and characterization of a demo turbine device to demonstrate the concept of microscale radial flow multistage turbomachinery and to provide design basis for development of the related technology. In the first part of this paper, the design approach for the microturbine with a planar geometry will be explained followed by aerodynamic analysis of blade rows at low Reynolds number using computational fluid dynamics (CFD). The configuration and fabrication of the demo device and the working principle of the components will be shown briefly for completeness. Finally, the characterization and test results of the rotating system and the microturbine performance will be presented.

2 Design Principles of Multistage Microturbines

2.1 Design Space for Microturbomachinery. Typically, large scale turbomachines operate at high Reynolds numbers (on the order of 10^6) and exhibit turbulent flow. Microscale configurations considered to date are mostly in the low Reynolds number range (100<Re<10,000), suggesting mainly laminar flow and higher viscous losses [1,2]. Unfortunately, the body of literature on blade passage aerodynamics (such as design correlations) is limited to high Reynolds numbers; hence new investigations are required at smaller scales.

Furthermore, the microfabrication approach constrains the designer to nontraditional configurations. Lithography allows precise patterning of the aerodynamic profiles on the surface of a silicon wafer, and these airfoil shapes are then transferred into the silicon substrate by deep reactive ion etching (DRIE). As illustrated in Fig. 1, this approach allows the creation of large arrays of well-defined blades that extend from the silicon substrate, which are most amenable to radial flow. An important outcome of this approach is that the flow area, A_f, increases linearly with radius, A_f=2πrh, since the blade height, h, is defined during a single etch step and is therefore constrained to be uniform. Due to this fabrication approach, each stage operates at a different tangential speed proportional to radius, U=Dr, and the blades are constrained to 2D extruded shapes, without twist along the span.

2.2 Radial Multistage Design Approach. The design approach consists of a meanline analysis based on velocity triangles with loss, blockage, and deviation factors. This low order modeling approach is the basis for preliminary design of traditional multistage turbomachinery [6]. The flow is, however, purely radial through concentric rotor and stator stages of constant blade height. In this paper, the flow is considered to be compressible (ideal gas)
and adiabatic. The nomenclature used to define the flow velocity components and the corresponding thermodynamic properties is illustrated in Fig. 2 for a stage composed of a stator and rotor. The process consists of conserving total enthalpy in a stator blade row or rothalpy in a rotor blade row, conserving mass, applying a loss coefficient correlation to define the total pressure, and applying a deviation correlation to define the exit flow angle. In addition, the blockage is applied to define the flow area in the mass conservation relation.

**Stator.** The pressure loss coefficient for a turbine stator is defined as

\[ Y_N = \frac{P_{01} - P_{02}}{P_{02} - P_{02}} \]

where the subscripts represent inlet (1), exit (2), and total (0) properties in the stationary frame. At the exit, the isentropic relation for the compressible flow is

\[ \frac{P_{02}}{P_2} = \left( 1 + \frac{k-1}{2} \frac{M_2^2}{k} \right)^{\frac{k}{k-1}} = C_2 \]

\[ P_0 = C_2 P_2 \]  

where the Mach number is determined from mass conservation at the exit, as shown in Eq. (3).

\[ M_2 = \frac{V_2}{\sqrt{kRT_2}}, \quad V_2 = \frac{m}{2\pi r_p h K \cos \alpha_2} \]

For mass conservation calculations, the density and mass flow rate are assumed at first. The exit velocity is calculated from mass conservation using velocity triangles to define the flow angles and considering the blockage.

The blockage was defined as the ratio of the effective flow area versus the geometric area \( K \). The exit flow angle, \( \alpha_2 \), is defined as the blade angle minus deviation, \( \delta \).

The exit temperature is determined from the energy conservation for the adiabatic flow in the stator based on

\[ T_2 = T_1 + \frac{1}{2C_p} \frac{V_1^2}{2C_p} - \frac{1}{2C_p} \frac{V_2^2}{2C_p} \]

\[ V_1 = \frac{m}{2\pi r_p h \cos \alpha_1} \]

When Eq. (2) is substituted into Eq. (1), the static pressure at the exit is expressed as a function of the inlet total pressure and the loss coefficient:

\[ P_2 = \frac{Y_N}{Y_N(1 - 1/C_2) + 1/C_2} \]

where the inlet total pressure is calculated from the given inlet static pressure and temperature and the assumed mass flow rate,

\[ P_{01} \left( 1 + \frac{k-1}{2} \frac{M_1^2}{k} \right)^{\frac{k}{k-1}} = \frac{V_1}{\sqrt{kRT_1}} \]

Using the ideal gas relation

\[ P_2 = \rho RT_2 \]

pressure in Eqs. (6) and (7) can be equated:

Fig. 1 Scanning electron microscopy (SEM) image of a typical radial multistage microturbine formed by DRIE, showing the rotors (left) and stators (right) on separate chips as well as a close-up view of one blade row (upper right). The turbine is assembled by laying the stator chip over the rotor chip in order to interdigitate the concentric blade rows.

Fig. 2 Velocity triangles in one turbine stage: (a) velocity triangle diagram and (b) h-s diagram
An iterative approach is then used to solve Eqs. (3)–(8) by changing the exit density until Eq. (8) is satisfied.

**Rotor.** In the rotor, the terms are redefined according to the relative coordinate and accounting for the centrifugal forces. The subscript 0 is replaced by 0w. The absolute velocity, $V$, is replaced by the relative velocity, $W$.

In the rotating reference frame of the rotor, centrifugal forces will do work on the flow and therefore relative total enthalpy is not conserved. Instead, rothalpy, $I_{0w}$, which represents relative total enthalpy corrected for the centrifugal work, is the conserved quantity along a steady adiabatic streamtube in the rotating reference frame [7]:

$$I_{0w} = H + \frac{W^2}{2} - \frac{U^2}{2}$$

(9)

Equation (4) therefore changes to

$$T_3 = T_2 + \frac{W^2 - U^2}{2C_p} - \frac{W^2 - U^2}{2C_p}$$

(4')

Accordingly, the aerodynamic pressure loss, $Y_R$, must be defined with respect to the isentropic total pressure, which also changes across the blade row due to the centrifugal work. Using the isentropic relations, we can define the equivalent conserved pressure as the rotary stagnation pressure, $P_{0w}$:

$$\frac{P_{0tw}}{P_3} = \left(1 + \frac{k-1}{2} \frac{M_{0t}^2}{2C_p} \right)^{\frac{k}{k-1}} = C_{0w}$$

(2')

The loss coefficient in the rotor, $Y_R$, is then redefined as

$$Y_R = \frac{P_{0tw} - P_{0tw}}{P_{0tw} - P_3}$$

(1')

where the numerator is the change in the rotary stagnation pressure across the blade row.

The process to find the exit condition in the rotor is the same as the stator except for the above replacements. Through this process, all the exit conditions are calculated for the assumed flow rate, which is the same through all stages. The flow rate is adjusted by iteration until the exit static pressure at the final stage matches the desired value. Power produced by the turbine is obtained from the total enthalpy difference between inlet and outlet multiplied by the mass flow rate. For the microturbine system design, this calculation process is repeated to obtain the steady operating rotor speed ($\omega$), which is obtained when the turbine power matches the power consumed through viscous drag by the surface area of the other parts.

2.3 Stage Configuration and Matching. For a turbine, work should be extracted from each stage with a similar loading distribution. The current radial outflow configuration results in the flow area and tangential speed that increase linearly with radius. Due to the work extracted from the flow, the pressure and density of the working fluid in the compressible flow decrease with radius. With the proper radial location of each stage, the density decrease can directly compensate for the through flow area increase and lead to approximately constant radial velocities in all stages. High velocities and hence power densities can therefore be maintained throughout the turbomachine. The desired layout should be such that the radius ratio between two locations is the inverse of the density ratio between those locations: $r_1/r_2 = \rho(r_2)/\rho(r_1)$. This leads to an outward flow turbine configuration, which is the opposite of normal practice. The centrifugal forces therefore pump the flow as we extract the work. For our typical operating conditions and multistage configuration, however, this opposing effect was found to be minimal compared with the advantage of operating all the stages at nearly constant high flow velocities. Given the constant blade height and the large radius ratio between the first and the last stage, we were forced to choose an outward flow configuration.

For a baseline mass flow of 24 mg/s, a power level of 1 W per stage is expected, which corresponds to 40 kW/kg. In order to match the specific power levels defined in the cycle analysis of a Rankine steam turbine, 10–30 stages are expected to be required [5]. However, Fig. 3 suggests that it is difficult to design more than five stages on a single rotor. In the six stage design, it is noticed that most stages have relatively flat curves, suggesting good robustness, except the last stage (sixth stage). Although its power output is more than any other stage, it occurs over a narrow range and drops dramatically at higher speeds to significantly negative values. The flow velocity can also change significantly and in some cases can exceed the critical velocity. This implies that if the flow rate, heat transfer, and/or the fluid density change, the power output can change drastically. By changing the geometry of the sixth stage of the turbine, this problem can be avoided. However, in a stable condition, the last stage does not produce as much power and is better to be removed. Therefore, a single rotor is expected to provide on the order of 5 W of mechanical power (200 kW/kg). Preliminary designs were also developed for a 28 W (1150 kW/kg), which consists of five individual rotors in series with power levels ranging from 3.8 W to 8.4 W. It operates with an inlet pressure of 8 MPa and a temperature of 780°C [8].

3 Blade Passage Aerodynamics

Three main parameters are required as input for the above design process: the loss coefficient, deviation, and blockage. These aerodynamic parameters depend on the blade passage geometry.
and operating conditions. The geometry is defined by the airfoil profile, the stagger angle, and the solidity (i.e., the ratio of the blade chord to spacing, $s/c$). Typically, correlations for loss and deviation are derived from experimental measurements and used in the initial design process. For a given geometry, they are found to depend on the incidence angle and Mach number of the incoming flow but are not a function of Reynolds number for traditional scale turbomachinery ($Re \sim 10^6$) [6]. In the operating regimes of microturbomachinery ($100 \leq Re \leq 100,000$) [9,10], the influence of Reynolds number is, however, expected to become important. As a first step, we have therefore chosen to use numerical simulations to explore the flow behavior in microturbomachinery cascades and to extract the main performance parameters (loss coefficient and deviation). These will enable the design of experiments to later validate the simulation results. This approach was mainly chosen since the size of microfluidic devices precludes the use of traditional instrumentation and requires the development of embedded sensors, not currently available. Scaled test apparatus have also been proposed and exhibit unique experimental challenges [11]. Fortunately at the microscale, flows are dominantly laminar and CFD solutions are expected to be increasingly accurate compared with large scale turbulent flows.

The current work is limited to the study of loss and deviation as a function of incidence and Reynolds number; the effect of Mach number will be studied subsequently since it was kept low in the current study and experiments ($M_{inc} \leq 0.2$). Two blade passage geometries are considered, as described next.

### 3.1 Blade Passage Geometry

The geometry chosen for this study is based on the NACA A3K7 turbine airfoil [12]. The primary series A3K7 is for reaction blades in which there is acceleration through the cascades. The camber line shape gives rapid turning in the forward part of the blade, where the Mach numbers are low. The profile is defined by a series of points for the camber line $(x_c,y_c)$ and a thickness distribution along that line (with a maximum thickness to chord of $t_{max}/c=20\%$). The stagger (angle of the blade to the radial direction) and camber (difference between leading and trailing edge angles) are adjusted in order to match the flow angles defined during the previous meanline analysis. Different levels of camber are achieved by scaling the tangential coordinate $(x_c,C \times y_c)$, and the blade is tilted to match the incoming flow angle. High camber (Rotor 1) and low camber (Rotor 3) blades are analyzed here, with configurations summarized in Table 1. The nominal solidity is $s/c=2$. The trailing edge is slightly modified in order to accommodate microfabrication limitations: It is shortened by 2% chord and kept to a minimum thickness of 4 μm.

### 3.2 Numerical Simulations

Commercial CFD software (FLUENT 6.1) is used for the numerical calculations. The steady-state Navier–Stokes equations are solved for the compressible laminar flow through a 2D section of the flow field near midspan with adiabatic walls. The blockage due to hub and shroud boundary layers is currently neglected, such that the flow passage is assumed to be of constant effective height. A segregated implicit solver is chosen with the SIMPLEx pressure-flow coupling algorithm and second order upwind schemes for the energy, momentum, and density equations. Cartesian grids were defined with 27,765 nodes (approximately $400 \times 60$) for Rotor 1 and 20,645 nodes (approximately $320 \times 60$) for Rotor 3, as shown in Fig. 4. The meshes were refined until the main parameter, which is the pressure loss coefficient, does not depend on the grid density for given boundary conditions. Because the calculations were done for low Reynolds numbers, leading to relatively thick boundary layers, the mesh density near the surface did not need to be as refined as for typical high Reynolds number calculations.

The working fluid was air with viscosity defined by Sutherland's law and with the following nominal operating conditions: inlet Mach number $M_1 = 0.14$, inlet total temperature $T_{01} = 300$ K, and exit static pressure $P_2 = 1$ atm. The inlet static pressure is adjusted to maintain $M_1$ constant as the incidence or Reynolds number is changed. Calculations were done in a stationary reference frame in order to simulate cascade test conditions. The computational domain was limited to a single blade passage, with inlet and outlet regions extending one chord upstream and downstream, respectively.

### 3.3 Effect of Incidence

Incidence, defined as the angle between the incoming flow and the blade leading edge, is a key parameter affecting the performance of microturbomachinery blades. In the current study and experiments, the incidence will be studied subsequently since it was kept low in the current study and experiments ($M_{inc} \leq 0.2$). Two blade passage geometries are considered, as described next.

![Typical computational grid. Upstream and downstream areas are not shown and the blade passage is repeated for illustration purposes: (a) Rotor 1 and (b) Rotor 3](image-url)

**Table 1 Blade row configurations**

<table>
<thead>
<tr>
<th>NACA A3K7</th>
<th>Rotor 1</th>
<th>Rotor 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord length</td>
<td>98.29 μm</td>
<td>114.76 μm</td>
</tr>
<tr>
<td>Chord axial</td>
<td>93.07 μm</td>
<td>89.53 μm</td>
</tr>
<tr>
<td>Inlet angle</td>
<td>56 deg</td>
<td>22 deg</td>
</tr>
<tr>
<td>Outlet angle</td>
<td>−74 deg</td>
<td>−60 deg</td>
</tr>
<tr>
<td>Stagger</td>
<td>19 deg</td>
<td>39 deg</td>
</tr>
</tbody>
</table>
parameter that affects the aerodynamic performance of macroscale blade rows. The first set of calculations explores the effect of the inlet flow angle on the loss coefficient (Eq. (1)) and the exit flow angle (i.e., deviation). Numerical results are shown in Fig. 5 for both geometries studied here (Rotors 1 and 3). The exit total pressure and exit flow angle are taken as the mass-average values at the outlet boundary (i.e., one chord downstream of the trailing edge); this approach leads to a conservative mixed-out loss coefficient. Within the blade passage, the Mach number reaches 0.5 \(< M_{\text{max}} < 0.6$.

The main observations in Fig. 5 are the increase in loss coefficient and reduction in flow turning with increasing incidence. The effect is most important for the high camber blade row (Rotor 1), whereas the low camber blade row (Rotor 3) remains practically unaffected. Unlike high Re turbomachinery, which exhibits flow separation at high positive or negative incidence angles, flow separation was not present in the low Re simulations. Negative incidences even slightly reduced the loss coefficient instead of increasing it.

### 3.4 Effect of Reynolds Number

The second set of calculations explored the effect of Reynolds number on the loss coefficient and the exit flow angle. Here, Re is defined with blade inlet properties and blade chord length. Numerical results are shown in Fig. 6. During these calculations, the inlet Mach number was maintained at $M_1 = 0.14 \pm 0.02$ and the incidence was fixed to zero. The Reynolds number was varied by scaling the model and changing the inlet total pressure to maintain $M_1$.

The most noticeable trend is the gradual increase in total pressure loss and reduction in flow turning as the Reynolds number is reduced. Below a critical Reynolds number of $Re_{\text{crit}} = 200–300$, the loss coefficient and deviation start to increase dramatically. This behavior will be discussed later, but it should be noted that no flow separation was observed. As the Reynolds number increases, the loss coefficient and exit flow angle tend to asymptote and therefore become less a function of Reynolds number, as expected.

For design purposes, the following loss coefficient correlation is proposed:

$$ Y = \frac{D_1}{\sqrt{Re}} + \frac{D_2}{Re} \quad (10) $$

The first term is inspired from the drag of a laminar boundary layer over a semi-infinite flat plate, while the second term stems from the consideration of the finite plate or airfoil length [13]. By fitting this expression to the numerical results, the following coefficients are found for the specific conditions listed in Secs 3.1 and 3.2: (a) Rotor 1: $D_1 = 6$, $D_2 = 400$ and (b) Rotor 3: $D_1 = 6$, $D_2 = 400$.
$D_2 = 170$. In general, the coefficients $(D_1, D_2)$ will depend on blade geometry (profile, camber, and solidity) and operating conditions (incidence and Mach number).

3.5 Discussions

3.5.1 Critical Reynolds Number. The critical Reynolds number $Re_{\text{crit}} = 200 – 300$ identified above appears to stem from the change in flow profile at the exit of the blade row, going from the boundary layer flow to the fully developed flow as the Re decreases. Total pressure contours are shown in Fig. 7 for two operating conditions: (a) $Re < Re_{\text{crit}}$ and (b) $Re > Re_{\text{crit}}$ to illustrate the boundary layer development along the blade passage. As better shown in Fig. 8, the profile of the velocity magnitude across the blade passage at the trailing edge (from suction to pressure side) shifts from the core flow at high Re to the merged boundary layers at low Re. This change in the regime appears to be associated with increased loss and reduced flow turning.

A first-order estimate of the critical Reynolds number can be derived from laminar flow relations. First, considering the Blasius solution for the laminar flow over a flat plate with no pressure gradient, the boundary layer thickness at the trailing edge can be defined as

$$\delta_i = 5.0 \sqrt{\frac{c}{u}} = \frac{5.0c}{\sqrt{Re}} \quad (11)$$

Since the boundary layers will merge at the trailing edge when $\delta_i = s/2$ and solidity is defined as $\sigma = c/s$, we can solve for $Re$:

$$Re_{\text{crit},1} = 100\sigma^2 \quad (12)$$
For the current configurations, \( \sigma = 2 \), hence \( \text{Re}_{\text{crit},1} = 400 \). A similar estimate can be done by considering the blade passage as a channel and defining the \( \text{Re} \) required such that the entry length, \( x_{\text{entry}} \), is equal to the blade chord, \( c \) [13]:

\[
\frac{x_{\text{entry}}}{D} = \frac{c}{s} = 0.04 \frac{uD}{v} = 0.04 \frac{uc}{v} = 0.04 \frac{\text{Re}}{s} = (13)
\]

Hence,

\[
\text{Re}_{\text{crit},2} = 25\sigma^2 \quad (14)
\]

and \( \text{Re}_{\text{crit},2} = 100 \) for \( \sigma = 2 \). These values appear to bound \( \text{Re}_{\text{crit}} \) from the numerical simulations. Based on these observations, one can suggest the following correlation for the critical Reynolds number:

\[
\text{Re}_{\text{crit}} = 60\sigma^2 \quad (15)
\]

### 3.6 Effect of Solidity

The above expressions suggest that blade rows of higher solidity would incur higher losses and deviation at low Reynolds number. Figure 9 illustrates this point by superposing results for \( \sigma = 2 \) and \( \sigma = 2.5 \). We can observe a ratio of approximately \((2.5/2)^2 \sim 1.5\) between the \( \text{Re}_{\text{crit}} \) (i.e., knee of the curve) for both solidities.

### 3.7 Effect of Blockage

Blockage is caused by many flow effects such as boundary layer on the endwalls and the blades, secondary flows, and tip leakage. In order to simplify the analysis and due to lack of relevant data at this small scale, only boundary layer on the endwalls was included in the calculations. The blockage was defined based on the boundary layer growth theory as follows:

\[
K = 1 - a\sqrt{\Delta r} \quad (16)
\]

where \( \Delta r \) represents radial position from the flow inlet and the constant \( a \) depends on the geometry and flow condition.

### 4 Configuration of Silicon Microturbine Device

In order to prove the design concept of the multistage microturbomachinery, a demo microturbine was developed and fabricated using the silicon microtechnology, which is commonly used in the semiconductor industry. The device was designed to have four stages of blade rows producing about 1 W of mechanical power each from steam with 200 °C and 3 atm at \( 1.3 \times 10^6 \) rpm, which corresponds to 270 m/s of the rotor tip speed [8].

The fabricated device encloses a 4 mm diameter rotor and consists of the flow paths to the turbine and bearings, as shown in Fig. 10. The bearing system, which supports the rotor axially and laterally as it rotates, is composed of hydrostatic thrust and journal bearings (JBs). The JB, which corresponds to the circumferential gap surrounding the rotor, exploits the loss mechanisms at the entrance of the narrow channel and difference of pressure distribution between opposite sides of the bearing to create a restoring force [8]. The thrust bearing (TB), which consists of small circular nozzles underneath the rotor, utilizes the flow resistances between nozzles and a planar clearance to balance out the axial movement of the rotor. The rotor is a disk with four concentric blade rows on one side, each composed of 80–180 blades. Four rows of stator blades also extend out from the upper layer, between the rotor blade rows, making an interdigitated radial flow turbine, as shown in Fig. 11. The height of the rotor blades is 70 \( \mu \text{m} \) and that of the stator blades is 50 \( \mu \text{m} \). The profile details of the fabricated blades and their shapes are described in Table 2 and Fig. 12, respectively. The geometry chosen for this study is based on the NACA A3K7 turbine airfoil, as in Sec. 2 [12]. The solidity of each blade row (chord-to-pitch ratio) is 2, and the radial gap between the blade rows is 25 \( \mu \text{m} \). The base tip clearance of the rotor blade is 1.5 \( \mu \text{m} \) and that of the stator is 20 \( \mu \text{m} \), which stems from the excessive etching during the fabrication process [8].

As illustrated in Fig. 10, the high pressure flow is fed from the top, does a right angle turn into the first blade row, and flows outward through the subsequent blade rows. Expansion of the working fluid provides the energy to drive the rotor. The increas-

### Table 2 Specifications of the stator and rotor blade rows

<table>
<thead>
<tr>
<th>Stage</th>
<th>Inlet angle (deg)</th>
<th>Outlet angle (deg)</th>
<th>Radial length (( \mu \text{m} ))</th>
<th>Chord length (( \mu \text{m} ))</th>
<th>No. of blades</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Stator</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>14</td>
<td>67</td>
<td>96.4</td>
<td>140.8</td>
<td>87</td>
</tr>
<tr>
<td>2</td>
<td>53</td>
<td>70</td>
<td>94.1</td>
<td>98.9</td>
<td>139</td>
</tr>
<tr>
<td>3</td>
<td>37</td>
<td>62</td>
<td>93.8</td>
<td>106.9</td>
<td>158</td>
</tr>
<tr>
<td>4</td>
<td>16</td>
<td>57</td>
<td>92.2</td>
<td>116.3</td>
<td>168</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Rotar</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>53</td>
<td>70</td>
<td>90.3</td>
<td>147.6</td>
<td>152</td>
</tr>
<tr>
<td>2</td>
<td>37</td>
<td>62</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>16</td>
<td>57</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-9</td>
<td>65</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
ing flow area along the stages maintains a nearly constant flow velocity, which can prevent premature choking in the flow passages. The bearing flows are supplied from the bottom. The journal bearing flow goes to the journal bearing plenum (JBP) and joins the turbine exit flow to be discharged after passing through the narrow gap. The thrust bearing flow splits into two directions and is discharged to the journal bearing plenum and to the atmosphere through the exhaust channel beside the nozzles.

The complete device is composed of five wafers, as shown in Fig. 10: one Pyrex glass wafer (wafer A), one silicon-on-insulator (SOI) (wafer B), and three silicon wafers (wafers C–E). The glass wafer, which is the first layer, was machined by ultrasonic drilling. The other four wafers are fabricated using silicon micromachining techniques such as photolithography, reactive ion etching (RIE), deep reactive ion etching (DRIE), and plasma-enhanced chemical vapor deposition (PECVD) to create channels, nozzles, bearings, and turbine blades. The machined layers are then anodically bonded (between wafers A and B) and fusion bonded (between wafers C–E). Details about the bearing system design, the fabrication process, and the operating procedures are beyond the scope of this paper; they are presented elsewhere [8,14,15].

5 Experimental Testing and Analysis

5.1 Experimental Setup and Operating Procedures. The test setup is composed of a gas flow control system and measurement sensors adopted from the previous microturbine development [16]. The working fluid is air for the experiments. The gas test consists of six separate pressure sensors (OMEGA® PX4202) and five mass flow controllers (MKS® 1179A) for bearing and turbine operation, an eight-channel pressure sensor (Scanivalve® Zoc17IP/RPX-APC 100PSID) for turbine interblade row pressure measurement, and one optical displacement probe (Philtec® Model 6D) for the measurement of the rotational speed of the rotor.

The eight pressure sensors are connected to interblade row static pressure taps, which correspond to the exit points of each blade row. The turbine inlet pressure, which is the first stator inlet pressure, could not be directly measured due to the lack of space for the pressure tap in the device but was derived by assuming the inlet pressure in the modeling and calculating interblade row pressures. The inlet pressure is defined when the calculated first stator exit pressure closely matches the measured one. The device is operated by maintaining the thrust bearing flow rate constant with a fixed feed pressure. The turbine flow rate is then increased by regulating the differential pressure between the supply and exhaust of the turbine using the metering valves.

5.2 Measured Performance and Predictions. During operation, pressure, flow rate, and rotation rate data were obtained. Figure 13 shows the turbine flow rate as a function of differential pressure between turbine inlet and outlet (dP). The uncertainty of the flow rate comes from the measurement error of the sensor. Figure 14 shows the rotational speed in rpm as a function of the turbine flow rate. The maximum speed measured from the test was 330,000 rpm, which corresponds to 70 m/s in tip speed for the 4 mm diameter rotor. Up to this speed, no instability in the rotor operation was observed. Currently, the maximum rotational speed achieved has been limited largely due to device failures, stemming from bearing limitations and mechanical failure [8].

Prediction. The model calculates the speed and turbine flow rate using the estimated pressure differential as an input. The speed results from the balance between the turbine power and surface drag torques acting on the rotor. The power generated by the turbine is estimated as the product of the mass flow rate and enthalpy difference between inlet and outlet. Detailed calculation procedure of the speed and power is beyond the scope of this paper but can be found in Ref. [8].

In this calculation, the loss coefficient and blockage were included in the meanline analysis. The correlation for the profile loss of the turbine blades was obtained by the 2D CFD calculations and expressed as a function of Reynolds number, as mentioned in Sec. 3. The coefficients in Eq. (10) were only predicted for Rotors 1 and 3 using the CFD calculations. For the other blade rows, they were linearly interpolated according to their camber angles assigning higher values for bigger angles.

Based on the CFD data, the impact of incidence was found to be relatively small compared with Reynolds number effects. For simplicity here, the constants are considered as functions of geometry only, especially camber angle, assuming negligible incidence effects and low Mach numbers (M<0.2).

The loss may be split into several categories such as profile loss, endwall loss, secondary loss, and tip clearance loss [17]. The correlation mentioned above corresponds to the profile loss, which is associated with boundary layer growth over the blade profile, a
characteristic of the two-dimensional flow around the airfoil. The other losses are related to 3D effects of the turbine flow. In the current modeling, these 3D effects were not predicted but were considered by multiplying the profile loss coefficient in Eq. (10) by a factor greater than unity. A factor of 1.4 was found to fit the calculation to the measured data as closely as possible. This approach assumes that the total loss is proportional to the profile loss, as suggested in large scale turbine research [17] and micro-scale investigations [9]. Furthermore, the factor of 1.4 used here is similar to that extracted from the comparison between 2D and 3D CFD calculations done for other microturbo machines [9].

The coefficient $a$ in Eq. (16) for blockage in each blade row was fixed to 8 based on previously published CFD results, which were obtained for higher Reynolds number range [9].

Although CFD investigations (Sec. 3) suggest that the exit flow angle varies at low Reynolds number range, deviation effects were not included in this model due to the difficulty in generalization for different angles from the limited CFD data. These simplifications are also applied to the calculations for characterization and projected performance at higher speeds in Secs. 5.3, 6.1, and 6.2.

The modeling results show slightly higher values for the same flow rate compared with the experimental data. As mentioned previously, the model does not include the loss factors for changing incidence and high Mach number and simply accounts for 3D losses by applying a correction factor. This may contribute to the higher prediction of the model. Also, the excessive tip clearance of the stator could be a cause. The clearance between the stator blade tips and the rotor surface was about 40% of the blade height, which resulted from the device fabrication process. That portion of the turbine flow can pass through the stator blade rows without appropriate turning, which would in turn reduce the fluid-to-mechanical energy conversion in the rotor, hence resulting in lower speed and power per unit flow.

### 5.3 Turbine Characterization

The pressures between the blade rows were measured at each operating condition. Figure 15 shows the pressure distribution at one operating point. The trend of the other data points is typically similar to that illustrated here. Generally the mechanical power generated by the flow is proportional to the pressure drop across the rotor blades. The pressure distribution shows that there is higher pressure drop in the first rotors than the last; $\Delta P(R1) > \Delta P(R2) > \Delta P(R3/R4)$, which reflects power production in the same order, as shown in Fig. 16.

The power produced from each stage is around 0.1 W in average, totaling 0.38 W at the highest measured speed.

### 6 Discussion

#### 6.1 Loss and Efficiency

Unlike large scale turbomachinery, which operates usually at high Reynolds numbers ($10^7 < Re < 10^9$), this microturbine runs at very low Reynolds number ranging from $Re = 10^2$ to $10^3$. The viscous loss effects are clearly expected to become dominant in the microturbine. The turbine cascade CFD calculation for microscale turbines showed that the pressure loss is at least one order of magnitude higher than for conventional machines [17]. Figure 17 shows the averaged loss coefficient as a function of the averaged Reynolds number for the current geometry and flow condition, using the meanline model and correlations proposed herein. As expected, the loss decreases as Re increases.

The total adiabatic efficiency of the turbine, which is defined as the ratio of the actual total enthalpy differential to isentropic total enthalpy differential, reaches 35.7% at $Re$ = 266, as shown in Fig. 18. When the Reynolds number increases, the viscous loss coefficient decreases so the adiabatic turbine efficiency should increase, as shown in Figs. 17 and 18. They reveal the significant impact of the Reynolds number, especially in this low range, and the benefit of higher-speed operation for better efficiency.

#### 6.2 Prediction for Higher-Speed Operation

As shown in Sec. 6.1, the efficiency at the measured speed is still much lower than that of large scale turbines, which are normally over 80–90%. Although it is not expected that small devices can reach efficiency levels of large machines due to fundamental scaling [10], it is desirable to increase the turbine efficiency up to a certain level to achieve reasonable performance of the overall power production microsystem.

Figures 19 and 20 show the projected turbine efficiency and the power produced from each stage. In the calculation process, the
static pressure at the outlet is fixed at atmospheric conditions and the inlet static pressure varies. The speed and flow rate at the highest performance point are $1.1 \times 10^6$ rpm and 3500 SCCM (SCCM denotes cubic centimeter per minute) with a turbine pressure ratio of 3:1. The projected efficiency reaches up to 63% at $\text{Re}=674$ for a total turbine power of 3.2 W. The estimated efficiency appears to be acceptable considering the micro-Rankine power cycle analysis [5]. Therefore, high-speed operation over $1 \times 10^6$ rpm or $\text{Re}>600$ is recommended for the current design to achieve the required levels of performance.

These predictions suggest that the current multistage turbine configuration may need to be revised since the power production and efficiency level beyond this threshold, setting a minimal scale on the order of 100 $\mu$m for flow velocities in the hundreds of m/s. Microturbomachinery must therefore operate at high speed to achieve acceptable efficiencies.

A demo four-stage microturbine device was fabricated out of five wafers using photolithography, shallow and deep etching, and silicon and glass bonding techniques. The turbine rotor with 4 mm in diameter is supported by gas lubricated bearings and was spun up to 330,000 rpm, producing 0.38 W of mechanical power. The measured pressure distribution between the blade rows shows close agreement with the modeling results. The modeling suggests 35% of total adiabatic efficiency with an average $\text{Re}=266$ in the turbine. The efficiency at $1.1 \times 10^6$ rpm is estimated to reach 63%, producing 3.2 W of total power at $\text{Re}=674$.

Considering the unprecedented small scale of the blades and low Reynolds numbers, the power production and efficiency level at the recommended operating range are quite encouraging in the perspective of the whole Rankine cycle efficiency. This research is expected to provide a guideline on the smallest size and lowest Reynolds number range for practical microturbines and to contribute to the development of other types of multistage microscale turbomachinery, such as pumps and compressors.

**Acknowledgment**

This work was supported by the NASA Glenn Research Center, Alternate Fuels Foundation Technologies program (Contract Nos. NAS3-02118 and NAS3-03105) monitored by Dr. Glenn Beheim. The authors gratefully acknowledge this support. The fabrication work was performed in part at the Cornell NanoScale Science and Technology Facility (CNF), a member of the National Nanotechnology Infrastructure Network, which is supported by the National Science Foundation (Grant No. ECS 03-35765).

**Nomenclature**

- $a$ = blockage coefficient
- $c$ = chord length
- $C_p$ = heat capacity
- $D$ = channel height
- $H$ = enthalpy
- $h$ = blade height
- $I$ = 1othalpy
- $i$ = incidence
- $K$ = blockage
- $k$ = heat capacity ratio
- $M$ = Mach number

**Fig. 18** Estimated total adiabatic efficiency of the turbine as a function of averaged Reynolds number

**Fig. 19** Model prediction of the total adiabatic efficiency at extended Re range

**Fig. 20** Model prediction of the power production from each stage at higher differential pressure

7 **Conclusion**

In this paper, we have presented an approach for the design of radial flow multistage silicon microturbomachinery and validated it through experimental demonstration of a microturbine with four stages. Using a model based on the meanline analysis, it is suggested that a planar radial flow configuration is appropriate but that a limited number of stages can be implemented per rotor in practice because it is hard to match the loading across the stages. The main aerodynamic parameters required for the design process (loss coefficient and deviation) were defined using computational fluid dynamics of the laminar flow through standard blade profiles but in the low Reynolds number range ($100<\text{Re}<1000$). It was found that dramatic increases in loss and deviation occur below a critical Reynolds number of approximately $\text{Re}_{\text{crit}}=200–300$. This behavior is associated with merging of the boundary layers at the exit of the blade passage. To maintain acceptable efficiency, it is therefore preferable to limit the scale of microturbomachinery beyond this threshold, setting a minimal scale on the order of 100 $\mu$m for flow velocities in the hundreds of m/s. Microturbomachinery must therefore operate at high speed to achieve acceptable efficiencies.
\[ m = \text{flow rate} \]
\[ P = \text{pressure} \]
\[ R = \text{gas constant} \]
\[ Re = \text{Reynolds number} \]
\[ r = \text{radius} \]
\[ s = \text{space between two blades} \]
\[ T = \text{temperature} \]
\[ t_{\text{max}} = \text{maximum thickness of blade} \]
\[ U = \text{tangential speed} \]
\[ u = \text{entry flow velocity} \]
\[ V = \text{absolute flow velocity} \]
\[ W = \text{relative flow velocity} \]
\[ x_{\text{entry}} = \text{entry length} \]
\[ Y = \text{pressure loss coefficient} \]
\[ \alpha = \text{inlet flow angle} \]
\[ \beta = \text{relative flow exit angle} \]
\[ \delta = \text{deviation angle} \]
\[ \delta_t = \text{boundary layer thickness} \]
\[ \Delta r = \text{radial position from the flow inlet} \]
\[ \rho = \text{density} \]
\[ \omega = \text{rotational speed} \]
\[ \sigma = \text{solidity} \]
\[ \nu = \text{dynamic viscosity} \]

Subscripts

0 = total property
1 = inlet of the stator blade
2 = exit of the stator or inlet of the rotor blade
N = stationary frame
R = rotating frame
w = property in the rotating frame

References