Numerical Solutions for Ultra-Micro Wave Rotors (U\(\mu\)WR)

Florin Iancu *
Michigan State University, East Lansing, MI, 48824

Janusz Piechna †
Warsaw University of Technology, Warsaw, Poland

and

Norbert Müller ‡
Michigan State University, East Lansing, MI, 48824

Starting in 1995, with the MIT “Micro Gas Turbine” project, the mechanical engineering research world has explored more and more the idea of “Power MEMS”¹. Microfabricated turbomachinery like turbines, compressors, pumps, but also electric generators, heat exchangers, internal combustion engines and rocket engines have been on the focus list of researchers for the past 10 years. The reason is simple: the output power is proportional to the mass flow rate of working fluid through the engine, or the cross-sectional area and the mass or volume of the engine is proportional to the cube of the characteristic length, thus the power density (\(\text{Power/Mass}=L^{-1}\)). This is the so-called “cube square law”. Although in theory everything is perfect, the following investigations showed that there are many engineering challenges at microscale and the solutions found in the past half of century for large scale mechanical devices do not necessarily apply to the new design space. This paper studies the possibilities of incorporating a wave rotor to an ultra-micro gas turbine. It discusses the advantages of wave rotor as topping units for gas turbines, especially at microscale and proposes some designs of ultra-micro wave rotors. The numerical simulations of these wave rotors are presented, results obtained using FLUENT, a Computational Fluid Dynamics (CFD) commercial code.

Nomenclature

\begin{itemize}
  \item \(A\) = cross-sectional area
  \item \(C_{p_{\text{air}}}\) = specific heat for air
  \item \(e\) = internal energy
  \item \(f\) = friction coefficient
  \item \(f_x\) = body forces in x-direction
  \item \(\gamma_{\text{air}}\) = air specific heat ratio
  \item \(\eta_{\text{comb}}\) = efficiency of combustion
  \item \(L\) = channel length
  \item \(m\) = mass flow rate
  \item \(\nu\) = kinematic viscosity
  \item \(\Pi\) = pressure ratio
  \item \(p\) = pressure
  \item \(\Pi_{\text{WR}}\) = wave rotor compression ratio
  \item \(q\) = heat flux
\end{itemize}

¹ Research Assistant, Department of Mechanical Engineering, 2500 Engineering Building, AIAA Student Member.
² Assistant Professor, Institute of Aeronautics and Applied Mechanics, 24 Nowowiejska Str.
³ Assistant Professor, Department of Mechanical Engineering, 2455 Engineering Building, AIAA Professional Member.
I. Introduction

Ultra Micro Gas Turbines (UμGT) are expected to be the next generation of power source for any number of applications from propulsion to power generation, from aerospace industry to electronic industry. Investigated by only a few research groups all over the world (MIT2-7, Stanford University8,9, ONERA – France10,11, The University of Tokyo12,13) the UμGT have yet to prove their utility. One of the main problems is their low overall thermal efficiency and output, and increased losses due to miniaturization. Particularly, obtained compressor efficiencies have been as low as 40-50%14, reducing optimum pressure ratios down to about 2. A proposed solution for this problem is integrating a wave rotor to the system, either in parallel to the combustion chamber or, in case of an internal combustion wave rotor, replacing the combustion chamber. The overall pressure ratio and the turbine inlet temperature mainly determine the gas turbine performance. At microscale, it has been shown that the compressor efficiency decreases if the pressure ratio goes above 214. Using a wave rotor will increase the overall pressure ratio (see Fig. 1), keeping the baseline compressor pressure ratio constant or even decreasing it. The wave rotor integration is most effective if its compression and expansion efficiencies are greater than those of the turbomachinery components of the baseline engine. An initial study on the efficiency of the compression process in a wave rotor channel has shown that values of around 70% can be achieved at microscale15. For large scale wave rotors values around 83% are consistent throughout the available literature.

![Figure 1. Temperature-Entropy diagram for the baseline and wave rotor enhanced engines.](image-url)
It was shown that at such microscale, the wave rotor can have the highest efficiency for shock wave pressure ratios in the range of 1.7-2, assuming that the microfabrication can generate a smooth enough surface with a low friction coefficient. The results show that the efficiency depends not only on pressure gain across the shock wave traveling through the wave rotor channel, but also depends highly on the loss coefficient for the channel geometry. According to the here employed model that is applicable for all wave rotor sizes, shorter wave rotor channels with larger diameter let expect a higher compression efficiency of the wave rotor.

The wave rotor is a pressure exchanger that uses the concept of direct transfer of pressure between fluids. A basic wave rotor consists of a rotating drum with straight channels arranged around its axis. The drum rotates between two end plates each of which has a few ports or manifolds, controlling the fluid flow through the channels. The number of ports and their positions vary for different applications. Since through rotation, the channel ends are periodically exposed to the ports located on the stationary end plates, compression and expansion waves are initiated within the wave rotor channels. Therefore, unlike a steady-flow turbomachine, which either compresses or expands the gas, both compression and expansion are accomplished within a single component. Two four-port wave rotor schematics are present in the Fig. 2. One is in a through flow (TF) configuration, in which both flows travel in the same direction. The other one is in reverse flow configuration (RF), in which each flow (gas or air) exits the same side it enters. For the TF wave rotor, the channels of the wave rotor are initially filled with air coming from the compressor (“fresh air”), through the low pressure air port (LPA). When they come in contact with the gases coming from the combustion chamber, which have high pressure and temperature (HPG), a shock wave is formed which compressed the fresh air. This high pressure air (HPA) is then evacuated to the combustion chamber, while the remaining gases which now have a lower pressure are scavenged towards the turbine. They are called pre-expanded gases or low pressure gases (LPG).

![Figure 2. Wave patterns for a through flow (TF) and reverse flow (RF) wave rotors.](image-url)
The way in which the wave-rotor topping enhances the cycle at microscale often differs from that at larger scales. At a large scale, mostly the goal is either to increase the cycle overall pressure ratio and temperature ratio or to substitute the wave rotor for costly high pressure turbomachinery stages\textsuperscript{16}. The most significant performance gain has been found for engines with low compressor pressure ratio and high turbine inlet temperature\textsuperscript{17,18}

From the manufacturing point of view, adding a wave rotor is much easier at microscale than at macro scale. The fabrication processes at microscale are mainly two dimensional (etching, deposition, oxidation, bonding), heavily constraining the geometry of the components. The wave rotor can easily be etched in silicon due to its common extruded 2D shape. Additionally, in a regenerative way the wave rotor allows to harvest some of the significant amount of heat conducted away from the combustor through the structure, which is a severe problem for microfabricated gas turbines and also reduces the efficiency of the spool compressor considerably.

Specific costs are high for microfabricated prototypes, which is one of the reasons the research is progressing with a slow pace. Another reason is that diagnostics and measurement at microscale are difficult and even destructive. The newly developed CFD commercial codes are speeding up the research process. Numerical investigations can now be faster and cheaper than experimental ones. The later ones being necessary only after the numerical model has been thoroughly verified. In the field of shock waves simulated by available commercial codes, the literature is not too vast. Although the wave rotor is based on a relatively simple engineering idea, its simulation is rather difficult to achieve. Shock waves behave differently at microscale than at macro scale. For a given Mach number the resulting particle velocity is lower but the pressure is higher\textsuperscript{19}. The diffusive transport phenomena can no longer be neglected at microscale, and viscous stresses at the boundaries tend to deform the shock wave front. Also the heat conduction to the wall prevents the flow from remaining adiabatic. At small scale the pressure rise across the shock increases with the decrease in the $ReD/4L$ factor for a constant Mach number.

II. Wave Rotor Integration into $\mu$GT

Starting from a baseline engine similar to the one developed by the MIT research group, several wave rotor topped ultra-micro gas turbine configurations are proposed\textsuperscript{15}. The wave rotor works in a four-port configuration, namely each cycle is completed after a wave rotor channel was exposed to all four ports, two at each end. The overall aspect of the wave rotor may include a larger number of ports depending on the number of cycles per revolution.

Figure 3a shows a wave rotor added at outer diameter of the disk formed by compressor/turbine unit\textsuperscript{4}. The end plates with the ports at either side of the wave rotor can be etched in the same wafer as the stationary guide vanes. A second possible design implies using additional wafers allowing a multi layer rotor, as shown in Fig. 3b. The major challenge with this design is the perfect axial alignment of the compressor/turbine unit with the wave rotor, which may be achieved with the common laser aligning method. The flow connection from the compressor to the wave rotor may be viewed as a challenge in respect of keeping the pressure loss small. However, the equivalent diameter of this connection may be designed sufficiently large. Further, this may aid in isolating the compressor case from the combustor heat, by introducing a larger fluid barrier. A third design concept introduces a new idea in respect of having multiple wave rotors arranged circumferentially around the compressor/turbine unit, as shown in Fig. 3c. Similar to the first classic design, this design requires less wavers then the second design, which translates into lower fabrication cost. The challenge associated with this design is driving all wave rotors at appropriate speed which may be achieved by arranging the wave rotor ports in proper angles, so that the impulse of the fluid streams can be utilized.
A new and innovative design is proposed next\textsuperscript{20}. Instead of the traditional axial wave rotor, a radial one seems to be far more suitable for ultra-micro gas turbines. Its shape and overall surface to height ratio is ideal for microfabrication processes, most of them being 2D processes. Plus, the variable cross-sectional area has been proven to provide a more efficient shock wave compression\textsuperscript{21}. Thermodynamically, the radial wave rotor (wave disc) will be incorporated same as the axial one, in parallel with the combustor. Schematics of the mechanical design are shown in Fig. 4.

![Design of radial wave rotor enhanced microturbines](image)

Figure 3. Designs of axial wave rotors enhanced microturbines: a) “classical design”, b) “two-layer” design, c) “external design”.

Figure 4. Design of radial wave rotor enhanced microturbines.
III. Numerical Analysis of Ultra-Micro Wave Rotor

A. Flow Gasdynamics

As a first step in analyzing a wave rotor and its components, it is necessary to present some basic notions about one-dimensional fluid flow, the gasdynamics and thermodynamics of it together with some knowledge on modeling a compressible fluid flow traveling through a channel. Next subchapter will describe the gasdynamic aspect of the problem, while the following one will present the mathematical modeling.

Among the processes taking place inside the wave rotor cells (channels) the main one is the development of compression and expansion waves between gases of different pressures and temperatures. This will constitute the primary energy exchange that takes place inside the cells. Other forms of energy exchanges are: heat transfer between fluids traveling through the channels and walls and friction losses due to the high viscous aspect of the flow at microscale.

Some assumptions are taken into consideration when writing the governing equations of the flow inside the wave rotor channels. Since the ratio of cell length over the cell width or height is greater than 10, this problem is accepted as being one-dimensional and will be treated as such. The gases involved in the process are considered to be ideal and compressible. Since the perfect gas constant and specific heat ratio are almost equal for air and exhaust gas, it will be assumed in this chapter that all gases have the same properties:

\[ R = 287 \text{ J/kg·K} \]
\[ \gamma = 1.4 \]

Considering all the presented assumptions, unsteady flow of compressible gas in a one dimensional process is described by the following equations\(^{22,23}\).

**Continuity equation**

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} = 0 \]  

**Navier-Stokes equation**

\[ \frac{\partial (\rho u)}{\partial t} + u \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial p}{\partial x} - \frac{\partial}{\partial x} \left( \rho u \frac{\partial u}{\partial x} \right) = 0 \]

For frictionless flow, the Navier-Stokes equation is reduced to the Euler equation.

**Energy equation**

\[ \frac{\partial}{\partial t} \left( \rho \left( e + \frac{u^2}{2} \right) \right) + \frac{\partial}{\partial x} \left( \rho u \left( e + rac{u^2}{2} \right) \right) + \frac{\partial (\rho u)}{\partial x} \left( \frac{\partial w}{\partial x} - \rho q - \rho f_x u \right) = 0 \]

Where \( q \) is the heat flux between fluid and walls.

Several variations of these equations are found in the wave rotor literature, since each paper deals with certain aspects of the flow investigations. Among these, Weber\(^{24}\) in his guide of shock wave engine design. Paxson has developed a simplified model by analogy with a shock tube\(^{25}\), which he later has improved together with Wilson\(^{26-30}\). Also, Fatsis and Ribaud have studied the unsteady flow behavior\(^{31}\).

A simple, but very descriptive set of equations has been provided by Piechna in his book, “Wave Machines, Models and Numerical Simulation”\(^{32}\). He has taken into consideration wall friction, leakage between rotor and ports, heat transfer and variation of cross-sectional area and ignores body forces.

**Continuity equation**

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \rho u \frac{d(ln A)}{dx} \pm \frac{\dot{m}}{A} = 0 \]
Momentum equation

\[ \frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial p}{\partial x} + \rho u^2 \frac{d(ln A)}{dx} + \int p \frac{u|u|}{2D} \pm u \frac{\dot{m}}{A} = 0 \]  

(5)

Energy equation

\[ \frac{\partial (\rho e)}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (up)}{\partial x} + u(p + \rho e) \frac{d(ln A)}{dx} - \dot{\gamma} \frac{\dot{m}}{A} e = 0 \]  

(6)

Where

- \( A \): channel local cross-sectional area
- \( \dot{m} \): mass flow rate of leaks
- \( \frac{d(ln A)}{dx} \): variation of cross-sectional area along the length of channel
- \( e = \frac{u^2}{2} + \frac{1}{\gamma - 1} \frac{p}{\rho} \): internal energy of the flow

B. Modeling of Unsteady Compressible Flow

The modeling of unsteady flow in tubes or channels was initiated by the graphical method of characteristics in the 1960’s. Advances in computer technology brought a fresh start to numerical methods, so the Euler equations (Eqs. (1) to (3)) were solved by different explicit and implicit codes, some written specifically for these applications (“in-house” codes), other created as software packages for solving any kind of fluid dynamics problem (Computational Fluid Dynamics commercial packages).

1. Explicit vs. Implicit

Always, a first issue that comes in solving a computational fluid dynamics (CFD) problem is what solver to use. The debate explicit vs. implicit solution has dated from the beginnings of numerical simulations and it is still going on. There is no clear criterion to decide which solution should be used for each problem; in fact some problems may provide similar results indifferent of solver used.

In general, explicit methods compute the next time step solution directly based on the known current time information. It is therefore necessary to keep the time step within the limit of physics in order to obtain stable solution in time. The implicit methods evaluate the solutions at two or more time steps. In this way, one can improve the accuracy of the solution, in principle. There are many examples in the integration of an ordinary differential equation by using multi-steps.

The main problems which arise from using one or the other solution are numerical stability and numerical accuracy. For stability, implicit solution method, which is more complex, allows for a larger time step, than explicit solution. But, since it is more complex, it requires more computational effort so even if the time step is larger, the computational time is higher too. For explicit solution, it can be proved that the condition of accuracy is strong related to the stability condition. For implicit solution, a factor of relaxation (damping) is introduced, which will make the solution more stable, thus making possible the increase in time step, but in the same time the results will be less accurate.

As an example, for a wave propagating through tube, the implicit solution with high damping will capture only the steady-state results, while an explicit one permits the investigation of transient processes. To be able to use implicit method in this case, a very small time step has to be used.

For non-viscous flows, where solution is conditionally stable, the explicit method works best.

As stated previously, the debate between implicit and explicit solution has not ended, and for the case of unsteady waves traveling through a channel/tube both methods have been used. Horrocks\(^{33}\), Pekkan\(^{34}\) and Lee\(^{35}\), DeCourtye\(^{36}\) have used implicit methods; Welch\(^{37,39}\), Piechna\(^{40,42}\), Larosiliere\(^{43}\), Eidelmann\(^{44}\) and Paxson\(^{25,29}\) have used explicit, while Greendyke\(^{45}\) has used a combination of explicit and implicit model.
2. One-dimensional CFD code

A one-dimensional code was created suitable for prediction of a porting solution for three and four port wave rotors. The solver is based on the Lax-Wendroff scheme with corrected flux transport. This numerical method automatically introduces a corrective term in the solution, which can be approximated with the viscous term in the Navier-Stokes equation, commonly referred to as artificial viscosity. The model has as inputs the length of one channel and the properties of the ports (pressure and temperature) as well as port opening and closing times. Based on its outputs, pressure distribution, velocity distribution and density distribution, as well as mass flow rate through the channel, through an iterative process the optimum timing of the wave rotor can be found. This code, combined with basic knowledge of gasdynamics and wave rotor diagrams is a powerful device in wave rotor design. In Fig. 5 the pressure distribution for the high pressure part of the rotor, the low pressure part, and the entire wave diagram can be seen. In Fig. 6, the mass flow rates are compared for the left and right boundaries (the two ends of the channel). The opening and closing of the ports as well as the direction of the flow can be clearly identified. Based on this diagram and knowing the duration of each time step, the mass balance of the process can be checked.

![Figure 5. Pressure distribution generated by the 1D CFD code: a) high pressure part, b) low pressure part, c) entire wave diagram.](image.png)
3. Setting up GAMBIT

GAMBIT 2.1 was used as a pre-processor for creating the geometry, mesh and domains necessary for the CFD software FLUENT. The wave rotor channels were designed as a sequence of channels, after the unfolded view method which allows 3D objects to be projected onto 2D surfaces. For the mesh standard quad elements were used in a map and pave type distribution. The pave distribution was used to accommodate the transition from finer mesh along the interface to a coarser mesh towards the end of the ports (Fig. 7).

In the axial configuration, 2D simulation of a wave rotor was made as follows: the channels are sliding between stationary ports with interface pairs created between end of channels and ports (Fig. 7). The pair of interfaces works only if at least one of the interfaces is made out of a single edge. If multiple edges are used for both interfaces in the pair, than the property transfer is not realized between the surfaces in contact. So the situation of Fig. 7a is acceptable, but for the one in Fig. 7b a small area was added to the end of the channels which will transform all the edges from the end of the channel into one. This is actually a more accurate simulation, since this will account for the leakage between the rotor and the casing (which contains the ports). Another way of creating the interface is to join the two ports together by a small slit (Fig. 7c). The slits added to the interfaces are 1/100 of the channel length. They might be created even smaller with the purpose of realistic modeling, but then the mesh has to be finer, which in turn will generate slower results.

Figure 6. Mass balance of the flow traveling through the channel.
4. Setting up FLUENT

For simulating the behavior of the flow through an ultra-micro wave rotor, the commercial CFD package, FLUENT 6.1 was used. The following paragraph will describe the solver used for obtaining the results. The results will be presented later on.

A double-precision solver was chosen due to the nature of the problem. Since the problem is unsteady, the residuals of the Euler equations will tend not to be in the same order of magnitude. The double-precision solver allows for a twelve order of magnitude drop of the residuals. The coupled solver allows for both implicit and explicit methods to be used. The coupled solver solves the governing equations of continuity, momentum, and energy simultaneously as a set of equations. If the coupled solver is used together with the energy equation, the viscous terms (artificial viscosity) are added to the later automatically. The stability of the solution is given by the CFL criterion. Let $g$ be a flow variable. Its value after a time $t+\Delta t$, function of previous time step, $t$, can be described in a basic Taylor series.

$$
\frac{\partial g}{\partial t}_{\text{avg}} \cdot \Delta t + \cdots
$$

But $\Delta t$ cannot be arbitrary. It has to be less or equal than some maximum value, estimated by a stability analysis performed on a set of linear equations in order for the solution to converge. The CFL criterion says that $\Delta t$ must be less than equal to the time required for a sound wave to travel between two adjacent grid points$^{46,47}$.

Figure 7. Mesh and interfaces setup for 2D axial wave rotor investigations: a) single port – multiple channel interface, b) single port – multiple channel interface with additional gap, c) multiple port – multiple channel interface with gap.
But $\Delta t / \Delta x$ (distance between adjacent grid points) is proportional to the Courant number (or CFL number). Because of this coupling, reducing the grid size will increase the solving time, unless the Courant number is increased. But for stability, Courant number has its limits. For explicit solution it is set to 1 and can go up to 2, while for implicit, it is set to 5 and can go up to 100$^{48}$. For the coupled solver, this is the main stability control parameter.

When using laminar flow conditions, the viscosity is modeled as function of temperature, based on the Sutherland model $^{49,50}$. The solution controls are Courant number (which is automatically adjusted for convergence of solution) and type of accuracy determined by the solution scheme. First order upwind scheme has been used, which is a first order accuracy solution, which means the quantities at faces are determined by assuming constant values through out the cell, calculated in the center and averaged over the cell. This scheme is useful and accurate when the flow is aligned with the grid. Otherwise, first order convective discretization increases the numerical error (due to diffusion).

IV. Results and Discussion

Based on the MIT Micro Gas Turbine Project$^{51,53}$, and on initial thermodynamic calculations, a set of boundary conditions for the wave rotor was generated.

<table>
<thead>
<tr>
<th>Wave Rotor Pressure Ratio</th>
<th>$\Pi_{WR}$</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ambient Temperature</td>
<td>$T_0$</td>
<td>293 K</td>
</tr>
<tr>
<td>Specific Heat for Air</td>
<td>$Cp_{air}$</td>
<td>1.005 kJ/Kg · K</td>
</tr>
<tr>
<td>Polytropic Coefficient for Air</td>
<td>$\gamma_{air}$</td>
<td>1.4</td>
</tr>
<tr>
<td>Efficiency of Combustion</td>
<td>$\eta_{Comb}$</td>
<td>80 %</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Pressure [bar]</th>
<th>Temperature [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low Pressure Air (LPA)</td>
<td>1.96</td>
</tr>
<tr>
<td>High Pressure Air (HPA)</td>
<td>3.93</td>
</tr>
<tr>
<td>Low Pressure Gas (LPG)</td>
<td>1.62</td>
</tr>
<tr>
<td>High Pressure Gas (HPG)</td>
<td>3.53</td>
</tr>
</tbody>
</table>

Using the 1D code for one channel described above, and having the overall geometric dimensions of the wave rotor from the designs in Fig. 3a, a porting solution was found for the axial wave rotor. The next table presents this set of data.

<table>
<thead>
<tr>
<th>Port openings</th>
<th>Degree</th>
<th>Circ. Length [mm]</th>
<th>Time [s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>HPG open</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00E-06</td>
</tr>
<tr>
<td>HPG close</td>
<td>6.87</td>
<td>0.60</td>
<td>3.28E-06</td>
</tr>
<tr>
<td>HPA open</td>
<td>3.83</td>
<td>0.335</td>
<td>1.83E-06</td>
</tr>
<tr>
<td>HPA close</td>
<td>9.08</td>
<td>0.792</td>
<td>4.33E-06</td>
</tr>
<tr>
<td>LPA open</td>
<td>12.98</td>
<td>1.133</td>
<td>6.19E-06</td>
</tr>
<tr>
<td>LPA close</td>
<td>23.33</td>
<td>2.037</td>
<td>1.11E-05</td>
</tr>
<tr>
<td>LPG open</td>
<td>9.93</td>
<td>0.867</td>
<td>4.74E-06</td>
</tr>
<tr>
<td>LPG close</td>
<td>18.74</td>
<td>1.636</td>
<td>8.94E-06</td>
</tr>
</tbody>
</table>
The full model had a 12 cycles per revolution (Fig. 8a) configuration but due to enormous computational volume, a reduced model was created later. The reduced model had a one cycle per revolution configuration and kept the same dimensions of the channel, the same tangential speed and same boundary conditions (Fig. 8b). The data for the reduced model is presented in the following table.

### Table 3. Details of the reduced model.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length of channels</td>
<td>0.001 m</td>
</tr>
<tr>
<td>Radius of the rotor</td>
<td>0.00042 m</td>
</tr>
<tr>
<td>Number of cells (channels)</td>
<td>20</td>
</tr>
<tr>
<td>Cell width</td>
<td>0.00013 m</td>
</tr>
<tr>
<td>Rotational speed</td>
<td>4,200,000 RPM</td>
</tr>
<tr>
<td>Tangential velocity</td>
<td>183 m/s</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Port openings</th>
<th>Degree</th>
<th>Circ. Length [mm]</th>
<th>Time [s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>HPG open</td>
<td>0</td>
<td>0.000</td>
<td>0.00E-06</td>
</tr>
<tr>
<td>HPG close</td>
<td>81.88</td>
<td>0.600</td>
<td>3.28E-06</td>
</tr>
<tr>
<td>HPA open</td>
<td>45.68</td>
<td>0.335</td>
<td>1.83E-06</td>
</tr>
<tr>
<td>HPA close</td>
<td>108.09</td>
<td>0.792</td>
<td>4.33E-06</td>
</tr>
<tr>
<td>LPA open</td>
<td>154.53</td>
<td>1.133</td>
<td>6.19E-06</td>
</tr>
<tr>
<td>LPA close</td>
<td>277.85</td>
<td>2.037</td>
<td>1.11E-05</td>
</tr>
<tr>
<td>LPG open</td>
<td>118.33</td>
<td>0.867</td>
<td>4.74E-06</td>
</tr>
<tr>
<td>LPG close</td>
<td>223.18</td>
<td>1.636</td>
<td>8.94E-06</td>
</tr>
</tbody>
</table>

Figure 8. 3D model of axial wave rotor: a) full model with 12 cycles per revolution, b) reduced model with 1 cycle per revolution.

An initial study was performed to decide the type of solver to be used. The explicit solver with laminar flow conditions, the implicit solver with laminar flow conditions as well as implicit solver with inviscid flow conditions (using the coupled equation option that generates artificial viscosity) were compared. Since the time step is small enough for both explicit and implicit solution to converge, all three methods provided similar results. These results are presented in the Figs. 9 and 11. This proves that the boundary conditions effect is smaller compared to inertial
effects. The artificial (numerical) viscosity is triggered by the pressure gradients, while the real (physical) viscosity is activated by the velocity gradients. Since the velocity is induced by the force of the shock, which is the pressure ratio, it can be clearly seen why the two viscosities generate the same effect.

Through Flow/Reverse Flow

For both CFD models (TF & RF), the 1D in-house code was used to decide for the porting scheme and to predict the desired wave patterns. It appears though, that with the CFD model the HPA port is opening too soon, meaning that the speed of the rotor is too high. In Fig. 11 it can be seen the expected pressure wave pattern with two shocks and an expansion wave for the high pressure part and one shock and one expansion for the low pressure part.
The contour plots are very similar with those in Fig. 5, that were obtained by the 1D code. The temperature distribution (Fig. 12) shows that the shock heats up the fresh air, from 432K to 500K, which is still much less than the temperature of the hot gases that are around 1500 K. The advantage of the RF wave rotor is that hot gases exit on the same side as they enter, thus scavenging all the hot temperature fluid and keeping the rotor cooled at the other side, which can be utilized at microscale to reduce the heat impact to the compressor.

Figure 11. Pressure distribution for 2D simulation of axial wave rotor: a) RF model, b) TF model.

Figure 12. Temperature distribution for 2D simulation of axial wave rotor: a) RF model, b) TF model.

The velocity distribution in Fig. 13c demonstrates that this truly is a TF wave rotor, having all the inflows and outflows in the same direction. The Figs. 13a and 13b, presenting a RF wave rotor display a correct direction of the flow through the ports, with positive direction on the high pressure part and negative on the low pressure part. Some oscillations in the flow direction are noticed when channels are closed, but their magnitude indicate that they do not influence the flow on the ends of the channels. Reynolds numbers are below 400 in all channels and at all times, so the assumption of laminar flow holds.
Simulations of the radial wave rotor were performed only for a single channel rotating about the stationary ports. The advantage of the radial wave rotor is that the centrifugal forces can help both the compression and the scavenging process. Pressure and temperature plots are shown in Fig. 14.

Figure 13. Velocity distribution in an axial wave rotor: a) RF – positive direction flow, b) RF – negative direction flow, c) TF – positive direction flow.

Simulations of the radial wave rotor were performed only for a single channel rotating about the stationary ports. The advantage of the radial wave rotor is that the centrifugal forces can help both the compression and the scavenging process. Pressure and temperature plots are shown in Fig. 14.

Figure 14. One channel radial wave rotor with HPG and HPA ports: a) pressure distribution, b) temperature distribution.
3D results of a previously investigated model are presented next. The reduced model shows that a higher rotational speed is needed, since the shock wave reaches the end of the channel short before the high pressure exhaust port opens (Fig. 15).

V. Conclusion

Utilizing a wave rotor to improve the performance of an \( U_{\mu}GT \) appears to be a promising solution. Even if pressure ratio of the baseline engine is already optimized, the wave rotor can still enhance both the overall thermal efficiency and cycle specific work output if the wave rotor compression efficiency is higher than that of the baseline engine compressor. Adding a wave rotor also reduces the baseline compressor pressure ratio and the exit temperature of the compressor. Furthermore, this may reduce the compressor diameter and rotational speed which results in reduced mechanical and thermal stresses and relaxed design constraints. From the manufacturing point of view, adding a wave rotor is much easier at microscale than at macro scale because the wave rotor can easily be etched in silicon due to its common extruded 2D shape. Additionally, in a regenerative way the wave rotor allows to harvest some of the significant amount of heat conducted away from the combustor through the structure, which is a severe problem for microfabricated gas turbines and also reduces the efficiency of the spool compressor considerably.

Four possible designs for integrating a wave rotor in microfabricated gas turbines are introduced: three axial and one radial wave rotor. Based on documented wave rotor efficiencies at larger scale and subsidized by a gasdynamic model that includes wall friction, the wave rotor compression efficiency at microfabrication scale could be estimated with about 70% \(^{15}\), which is much higher than the obtained efficiency of compressors in a microfabricated gas turbine.

The preliminary CFD simulations confirm the theory of Brouillette \(^{19}\), showing a diffused shock wave (Fig. 11) over the length of the channel, and an increase in pressure ratio across the “shock”. The complete 2D FLUENT analyses confirm that the initial 1D design obtained with the in-house code is a valid design. But these commercial code simulations take enormous computational effort, so they are not suitable for initial geometry investigation. After the 1D code predicts an initial geometry, FLUENT may be used to fine tune the model by doing several design iterations. The CFD simulations verify the wave pattern as well as compression and expansion process predicted by theory. The fresh air was compressed by the hot gases to the desired value. Some mismatches were discovered regarding the speed of the rotor with respect to the speed of the compression and expansion waves. All the characteristics of an unsteady flow were well modeled by the CFD code and the results proved to be a valuable link in the design process of an ultra-micro wave rotor. Furthermore, after the geometry is set, FLUENT may be used for simulation of several cycles to investigate the behavior of the wave rotor in normal operating conditions.
References


