A Two-Dimensional Numerical Study of the Flow Inside the Combustion Chamber of a Motored Rotary Engine

T.I-P. Shih and S.L. Yang
University of Florida
Gainesville, Florida

and

H.J. Schock
Lewis Research Center
Cleveland, Ohio

Prepared for the
1986 SAE International Congress and Exposition
Detroit, Michigan, February 24–28, 1986
A Two-Dimensional Numerical Study of the Flow Inside the Combustion Chamber of a Motored Rotary Engine

T.I-P. Shih and S.L. Yang
University of Florida

and

H.J. Schock
NASA Lewis Research Center

1986 SAE International Congress and Exposition
Detroit, Michigan
February 24–28, 1986
The appearance of the code at the bottom of the first page of this paper indicates SAE's consent that copies of the paper may be made for personal or internal use, or for the personal or internal use of specific clients. This consent is given on the condition, however, that the copier pay the stated per article copy fee through the Copyright Clearance Center, Inc., Operations Center, 21 Congress St., Salem, MA 01970 for copying beyond that permitted by Sections 107 or 108 of the U.S. Copyright Law. This consent does not extend to other kinds of copying such as copying for general distribution, for advertising or promotional purposes, for creating new collective works, or for resale.

Papers published prior to 1978 may also be copied at a per paper fee of $2.50 under the above stated conditions.

SAE routinely stocks printed papers for a period of three years following date of publication. Direct your orders to SAE Order Department.

To obtain quantity reprint rates, permission to reprint a technical paper or permission to use copyrighted SAE publications in other works, contact the SAE Publications Division.
A numerical study was performed to investigate the unsteady, multidimensional flow inside the combustion chambers of an idealized, two-dimensional, rotary engine under motored conditions. The numerical study was based on the time-dependent, two-dimensional, density-weighted, ensemble-averaged conservation equations of mass, specie, momentum, and total energy valid for two-component ideal gas mixtures. The ensemble-averaged conservation equations were closed by a K-ε model of turbulence. This K-ε model of turbulence was modified to account for some of the effects of compressibility, streamline curvature, low-Reynolds number, and preferential stress dissipation. Numerical solutions to the conservation equations were obtained by the highly efficient implicit-factored method of Beam and Warming. The grid system needed to obtain solutions were generated by an algebraic grid generation technique based on transfinite interpolation. Results of the numerical study are presented in graphical form illustrating the flow patterns during intake, compression, gaseous fuel injection, expansion, and exhaust.

DIFFICULTIES WITH THE SUPPLY of aviation gasoline in several parts of the world have given rise to the need for more fuel-efficient and more fuel-tolerant stratified-charge rotary engines, it is necessary to have a good understanding of the physics (fluid flow, heat transfer, and combustion) that take place inside the engines' combustion chambers. Relatively few investigators have studied the physics taking place inside rotary engines. Experimental research in rotary engines so far has been confined to studies on engine performance and emissions (4–10) and to high-speed photographic studies of combustion (11,12). Mathematical research in rotary engines has been broader in scope but has been limited to quasi-dimensional (13–22) and one-dimensional (23,25) analyses. To date, no studies have reported details of the unsteady, multidimensional flowfield inside rotary engine combustion chambers.

The unsteady, multidimensional flowfield inside a rotary engine combustion chambers is complex. Owing to this complexity, progress towards understanding the details of this flowfield can only be made in a piecemeal manner by focusing on specific problems. Furthermore, it is necessary to first study simplified rotary engines and later study increasingly more realistic rotary engines.

The objective of this study was to use a numerical method to investigate the flow patterns inside one of the combustion chambers of a simplified rotary engine under motored conditions. The simplified rotary engine studied here was two-dimensional (i.e., it was a rotary engine of infinite width) with "planar" gaseous fuel injection into the combustion chamber during the compression process. Also it was assumed that there was no leakage across the seals. The present study constitutes the first attempt at numerically studying the unsteady, multidimensional flow inside rotary engines. This study was conducted for a motored configuration in order to compare model results to experimental data which will soon be available.

*Numbers in parentheses designate references at end of paper.
In this report the simplified rotary engine studied is described in detail. Afterwards, formulation of the problem along with the essence of the numerical method of solution are presented. Finally, results for one numerical simulation are presented illustrating the flow patterns during intake, compression, gaseous fuel injection, expansion, and exhaust.

DESCRIPTION OF PROBLEM

A motored, two-dimensional rotary engine is shown in Fig. 1. It consists of a housing, an intake port, an exhaust port, a gaseous fuel injector, a rotor, three seals, three combustion chambers (the three regions bounded by the rotor, housing, and seals), and a shaft. The equations that describe the geometry of this rotary engine are given in Refs. 26 and 27 and summarized in Table I. The Cartesian coordinates \( X_1 \) and \( Y_1 \) describing the inner surface of the housing (denoted as surface 1 in Fig. 1) are given by Eqs. (1) and (2) (Table I). The Cartesian coordinates \( X_2 \) and \( Y_2 \) describing the outer surface of the rotor (denoted as surface 2, 3, and 4 in Fig. 1) are given by Eqs. (3) to (12) (Table I). The geometry of the rotor pockets is given by Eqs. (10) to (12). Here, it is noted that the parameter \( C \), representing the distance between the rotor apex and the housing is set at 0.004 m which is somewhat large. The value chosen is to facilitate the numerical method of solution. Since leakage was not considered, the value chosen for \( C \) should not seriously affect the flow patterns. The origin and orientation of the Cartesian coordinate system is shown in Fig. 2(a).

For this motored rotary engine, as the rotor rotates inside the housing, each of the three combustion chambers goes through five processes in the following order: intake, compression, gaseous fuel injection, expansion, and exhaust. Because of this symmetry, we only need to study the physics taking place inside one of the three combustion chambers. The combustion chamber chosen is the region bounded by surfaces 1, 2, 5, and 6 in Fig. 1.

As mentioned previously, leakage across the seals was not considered. If leakage across the seals was considered, we could still study the physics of the airflow taking place inside the rotary engine by only analyzing one combustion chamber.

Initially, the combustion chamber to be studied is located at the beginning of the intake process (i.e., set \( \theta \) in Eqs. (3) and (4) to zero) and is filled with stagnant air at a constant static temperature \( T_1 \) and pressure \( P_1 \). Suddenly, the shaft starts to rotate at a constant angular velocity \( \Omega \) of 5000 rpm (revolutions/min) which in turn causes the rotor to rotate in the clockwise direction. This starts the intake process.

During the intake process, air enters the combustion chamber through the intake port. The stagnation temperature \( T_0 \) and pressure \( P_0 \) (\( P_0 > P_1 \), i.e., turbocharged) of the entering air are both constants. During the early part of the intake process, the exhaust port is also open. Thus some of the air also leaves the combustion chamber through the exhaust port. The back pressure of the exhaust port is maintained at \( P_{exh} \) (\( P_{exh} < P_1 \)). Once the trailing apex (denoted by \( \alpha_T \) in Fig. 1) passes point B on the housing (see Fig. 1), the intake port is closed. The reason for closing this port earlier is to facilitate the numerical method of solution. Once the intake port closes, the compression process starts. Later, between crank angles \( \theta_1 \) and \( \theta_2 \), a gaseous fuel (octane) is injected into the combustion chamber. Still later when the volume of the combustion chamber starts to increase, the expansion process starts. Once the leading apex (denoted by \( \alpha_L \) in Fig. 1) passes point C on the housing (see Fig. 1), the exhaust port opens initiating the exhaust process. The late opening of the exhaust port is to facilitate the numerical method of solution. As noted earlier, the back pressure of the exhaust port is maintained constant.

For the problem just described, we are interested in knowing the details of the flow patterns inside the combustion chamber during intake, compression, gaseous fuel injection, expansion, and exhaust.

FORMULATION OF THE PROBLEM

The equations that govern the problem described in the previous section are the conservation equations of mass, species, momentum, and total energy for two-component ideal gas mixtures. In this numerical study, the following: 

SHIH
TABLE I. - EQUATIONS DESCRIBING GEOMETRY OF ROTARY ENGINE

<table>
<thead>
<tr>
<th>Equation</th>
<th>Eq. No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>(X_1 = E \cos (3A) + (R + C) \cos (A))</td>
<td>(1)</td>
</tr>
<tr>
<td>(Y_1 = E \sin (3A) + (R + C) \sin (A))</td>
<td>(2)</td>
</tr>
<tr>
<td>(X_2 = E \sin (\theta) + X_{RO} \cos (\theta/3) + Y_{RO} \sin (\theta/3))</td>
<td>(3)</td>
</tr>
<tr>
<td>(Y_2 = E \cos (\theta) + Y_{RO} \cos (\theta/3) - X_{RO} \sin (\theta/3))</td>
<td>(4)</td>
</tr>
</tbody>
</table>

where

\[
\theta = \int_0^t \Omega \, dt = \Omega t
\]  

\[
X_{RO} = X_R \cos (\pi/6) + Y_R \sin (\pi/6)
\]  

\[
Y_{RO} = Y_R \cos (\pi/6) + X_R \sin (\pi/6)
\]  

\[
X_R = R \cos (2V) - (3E^2/R) \sin (6V) \sin (2V)
\]

\[
+ 2E \left[ 1 - \left( \frac{9E^2}{R^2} \right) \sin^2 (3V) \right]^{1/2} \cos (3V) \cos (2V) - P_x
\]  

\[
Y_R = R \sin (2V) - (3E^2/R) \sin (6V) \cos (2V)
\]

\[
+ 2E \left[ 1 - \left( \frac{9E^2}{R} \right) \sin (3V) \right]^{1/2} \cos (3V) \cos (2V) - P_y
\]  

\[
P_x = P \cos (2V)
\]  

\[
P_y = P \sin (2V)
\]  

\[
P = \begin{cases}
0 & \text{if } V_1 \leq V \leq V_T \\
\frac{P_1}{2} \left[ 1 - \cos \left( \pi(V_1 - V)/(V_1 - V_2) \right) \right] & \text{if } V_2 \leq V \leq V_1 \\
\frac{P_2}{2} + (P_1 - P_2)(V - V_3)/(V_2 - V_3) & \text{if } V_3 \leq V \leq V_2 \\
\frac{P_3}{2} \left[ 1 - \cos \left( \pi(V - V_4)/(V_3 - V_4) \right) \right] & \text{if } V_4 \leq V \leq V_3 \\
0 & \text{if } V_L \leq V \leq V_4
\end{cases}
\]  

\(aA\) and \(V\) are parametric parameters. \(A\) varies between 0 and \(2\pi\). \(V\) varies between \(V_L = \pi/6\) and \(V_T = 7\pi/6\) for surface 2 (see Fig. 1); between \(V_L = 5\pi/6\) and \(V_T = 7\pi/6\) for surfaces 3; and between \(V_L = 3\pi/2\) and \(V_T = 11\pi/6\) for surface 4. \(E = 0.015\,m, R = 0.1045\,m, C = 0.004\,m, P_1 = P_2 = 0.005\,m, V_L = V_L + 0.75(V_T - V_L), V_2 = V_L + 0.625(V_T - V_L), V_3 = V_L + 0.375(V_T - V_L), V_4 = V_L + 0.25(V_T - V_L), \) and \(\Omega = 5000\,rpm.\)

Simplifications were made to the conservation equations: (I) all thermodynamic and transport properties (except for the binary diffusion coefficient) were taken to be constants, (II) the binary diffusion coefficient was taken to be inversely proportional to the local mixture density, (III) pressure gradient contribution to specie diffusion and momentum transport due to species diffusion were neglected, (IV) irreversible coupling effects between the temperature gradient and the specie concentration gradient (Soret and Dufour effects) were neglected, (V) gravity force was not considered, (VI) bulk viscosity was taken to be zero, (VII)
radiation heat transfer was not considered, and (VIII) chemical reactions did not occur. Furthermore, since the flow of interest is turbulent and present computing capabilities prohibit direct or large eddy simulations of turbulence, the conservation equations were density-weighted, ensemble-averaged and closed by a K-ε model of turbulence (28). The K-ε model of turbulence employed was modified to account for some of the effects of compressibility (29), streamline curvature (30), low Reynolds number (31), and preferential stress dissipation (32).

The density-weighted, ensemble-averaged conservation equations and the K-ε model of turbulence used in this numerical study are summarized in Table II. Equations (13) to (44) in Table II constitute a closed system in seven dependent variables: density (ρ), mass fraction of air (X_A), x-component of the velocity (U), y-component of the velocity (V), total energy (E), turbulent kinetic energy (K), and dissipation rate of turbulent kinetic energy (ε).

The boundary and initial conditions corresponding to Eqs. (13) to (44) are given in Table III. The no-slip condition requires the fluid velocity next to a solid wall to be equal to the velocity of the solid wall. This condition gives Eqs. (45) and (46). Here, it is noted that the K-ε model of turbulence employed was modified to account for some of the effects of low-Reynolds number so that the model can be applied all the way to the wall (i.e., wall functions are not needed).

The gas temperature next to a solid wall equals the temperature of the solid wall. With all walls maintained at given temperatures, this boundary condition results in Eqs. (47) and (48).

The walls are assumed to be nonporous. This boundary condition is expressed by Eq. (49).

The turbulent kinetic energy and the dissipation rate of turbulent kinetic energy are both set equal to zero at solid walls because of the turbulence model employed. These two boundary conditions are given by Eq. (50).

The stagnation temperature (T_0) and pressure (P_0) of the air entering the combustion chamber through the intake port are constant as reflected by Eqs. (51) and (52). The mass fraction of the air entering the combustion chamber through the intake port is unity and is given by Eq. (53). At the intake port, the x-component of the velocity is zero and the y-component of the velocity is determined by the Bernoulli equation. These two boundary conditions are given by Eqs. (54) and (55). The turbulent kinetic energy and the dissipation rate of turbulent kinetic energy of the air entering through the intake port are given by Eqs. (56) and (57).

At the fuel injector, gaseous fuel (octane) is injected into the combustion chamber when the crank angle θ is between θ_1 and θ_2. During fuel injection, the stagnation temperature (T_f) is kept constant as shown by Eq. (58). The fuel is injected at an angle α into the combustion chamber (see Fig. 1). The x and y-components of the velocity are given by Eqs. (59) and (60). The first-derivative of the pressure normal to the fuel injector is taken to be zero. This boundary condition is given by Eq. (61). The turbulent kinetic energy and the dissipation rate of turbulent kinetic energy of the fuel at the fuel injector are given by Eqs. (62) and (63).

At the exhaust port, the back pressure (P_exh) is maintained constant. This condition is used to calculate the y-component of the velocity for the gas leaving the combustion chamber as indicated by Eq. (64). At the exhaust port, the x-component of the velocity along with the first-derivative with respect to y of the following variables are zero: mass fraction of air, pressure, temperature, turbulent kinetic energy, and dissipation rate of turbulent kinetic energy. These boundary conditions are given by Eqs. (65) and (66).

At time t equal to zero, the rotor is at the beginning of the intake process (θ = 0) and the combustion chamber is filled with air at static pressure P_i and static temperature T_i. At this time, the air is in a state of homogeneous turbulence with zero mean velocity. The turbulent kinetic energy and the dissipation rate of turbulent kinetic energy of the air are both constant. These initial conditions are specified by Eqs. (67) to (72).

NUMERICAL METHOD OF SOLUTION

Solutions to the governing equations formulated in the above section can only be obtained by numerical methods. In this study, solutions were obtained by a finite-difference method.

All finite-difference methods involve two approximations. First, the continuous domain of the problem must be replaced by time levels and a system of grid points. Second, the governing partial differential equations must be replaced by finite-difference equations. These two approximations are described below.

TIME LEVELS AND GRID SYSTEM - Finite-difference methods only provide solutions at the time levels and the grid points. The time increment between two successive time levels (i.e., the time-step size) must be made small enough to ensure numerical stability and temporal accuracy. For the present problem, the rotor is started impulsively from 0 to 5000 rpm. For such a problem, it is necessary to use a very small time-step size in the beginning (Courant number much smaller than unity). As time progresses, the time-step size used can be allowed to increase steadily until it reaches some preassigned value (corresponding to a Courant number much larger than unity). Such variable time-step sizes were employed here. The time-step size used between time levels n and n+1 (denoted by Δt_{n+1}) was as follows:
TABLE II. GOVERNING EQUATIONS

<table>
<thead>
<tr>
<th>Equation</th>
<th>Eq. No.</th>
</tr>
</thead>
</table>
| \[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x} (\rho \dot{u}) + \frac{\partial}{\partial y} (\rho \dot{v}) = 0
\] | (13) |
| \[
\frac{\partial}{\partial t} \rho \dot{x} + \frac{\partial}{\partial x} (\rho \mu_x) + \frac{\partial}{\partial y} (\rho \mu_y) = \frac{\partial}{\partial x} \left( \frac{\partial}{\partial x} \frac{\partial \rho \dot{x}}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{\partial}{\partial y} \frac{\partial \rho \dot{x}}{\partial x} \right)
\] | (14) |
| \[
\frac{\partial}{\partial t} \rho \dot{u} + \frac{\partial}{\partial x} (\rho \mu_x) + \frac{\partial}{\partial y} (\rho \mu_y) = \frac{\partial}{\partial x} \left( \frac{\partial}{\partial x} \frac{\partial \rho \dot{u}}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{\partial}{\partial y} \frac{\partial \rho \dot{u}}{\partial x} \right)
\] | (15) |
| \[
\frac{\partial}{\partial t} \rho \dot{v} + \frac{\partial}{\partial x} (\rho \mu_x) + \frac{\partial}{\partial y} (\rho \mu_y) = \frac{\partial}{\partial x} \left( \frac{\partial}{\partial x} \frac{\partial \rho \dot{v}}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{\partial}{\partial y} \frac{\partial \rho \dot{v}}{\partial x} \right)
\] | (16) |

where

\[
\ddot{t} = 2 \frac{\ddot{\nu}}{\ddot{t}} + \frac{\ddot{t}}{3} \nu + \nu
\] | (20) |
\[
\ddot{c} = 2 \frac{\ddot{\nu}}{\ddot{c}} + \frac{\ddot{c}}{3} \nu + \nu
\] | (21) |
\[
\ddot{c} = \ddot{t} \dot{c} + \ddot{t} \dot{c}
\] | (22) |
\[
\ddot{c} = \ddot{u} + \ddot{v}
\] | (23) |
\[
\ddot{c} = \dot{u} + \dot{v}
\] | (24) |
\[
\ddot{c} = \dot{u} + \dot{v}
\] | (25) |
\[
\ddot{c} = \dot{u} + \dot{v}
\] | (26) |
\[
\ddot{c} = \dot{u} + \dot{v}
\] | (27) |
\[
\ddot{c} = \dot{u} + \dot{v}
\] | (28) |
\[ \lambda_t = \nu_t \frac{C_p}{\rho_{t_t}} \]  

\[ \tilde{p} = \left\{ \begin{array}{l} e^{-\rho x_A} \left[ h_A^o - h_B^o \right] - \left( C_{PA} - C_{PB} \right) T_i \right\} - \rho \left( h_B^o - C_{PB} T_i \right) - \frac{1}{2} \rho \left( u^2 + v^2 \right) \\
- \rho K \left\{ \frac{R_{\rho x_A}}{M_A - M_B} + \frac{R}{M_B} \right\} \end{array} \right\} \]  

\[ \tilde{T} = \left\{ \begin{array}{l} e^{-\rho x_A} \left[ h_A^o - h_B^o \right] - \left( C_{PA} - C_{PB} \right) T_i \right\} - \rho \left( h_B^o - C_{PB} T_i \right) - \frac{1}{2} \rho \left( u^2 + v^2 \right) \\
- \rho K \right\} \]  

\[ G = \nu_t \left[ \frac{2}{2} \left( \frac{\partial v}{\partial x} \right) + \frac{2}{2} \left( \frac{\partial v}{\partial y} \right) + \left( \frac{\partial v}{\partial x} + \frac{\partial v}{\partial y} \right) \right] - \frac{2}{3} \nu_t \left( v \cdot y \right) \left( \nu_t \cdot y \right) + \rho K \]  

\[ S = C_1 f_1 K \left( \frac{C_1}{C_1'} + \frac{p_n}{p_s} \right) - C_2 f_2 \rho e^2/K + f_3 \]  

\[ P_n = \nu_t \left[ \frac{2}{2} \left( \frac{\partial u}{\partial x} \right) + \frac{2}{2} \left( \frac{\partial u}{\partial y} \right) \right] - \frac{2}{3} \nu_t \left( v \cdot y \right) \left( \nu_t \cdot y \right) + \rho K \]  

\[ P_s = \nu_t \left( \frac{\partial v}{\partial x} + \frac{\partial v}{\partial y} \right) \]  

\[ f_3 = \rho e (v \cdot y) + f_B \left( \frac{2 \nu_t}{\rho} \frac{\partial^2 u}{\partial y^2} \right)^2 \]  

\[ C_1' = \left[ C_1 + a (C_1 - 1.5) \right] / C_1 \]  

\[ C_1'' = \left( 3 + \frac{3}{2n} \right) / C_1 \]  

\[ a = \frac{3}{2} \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 \right] \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 \right] - 1 \]  

\[ n = \frac{3}{\sqrt{2a}} \]  

\[ D = f_B 2 \nu \left( \frac{\partial K}{\partial \rho} \right)^{1/2} \]  

\[ f_\mu = f_B f_\infty \exp \left[ -3.4 \left( 1 + R_T / 50 \right)^2 \right] \times \left[ 1 + 0.57 \frac{K^2}{\epsilon^2} \frac{u_s}{\epsilon^2} \frac{\partial}{\partial \tau} \right] \]  

\[ f_2 = f_B 2 \left[ 1 - 0.3 \exp \left( - R_T^2 \right) \right] \]  

\[ R_T = \frac{2}{\nu} \]  

**Table II - Continued.**
$\bar{z}$ = density-weighted mean of $z$, $\bar{g}$ = unweighted mean of $g$, $\bar{u}$ = density, $\bar{v}$ = x-component velocity, $\bar{V}$ = y-component velocity, $\bar{e}$ = total energy, $\bar{P}$ = pressure, $\bar{T}$ = temperature, $K$ = turbulent kinetic energy, $\epsilon$ = dissipation rate of turbulent kinetic energy, $\mu$ = dynamic viscosity, $D_{AB}$ = binary diffusion coefficient, $\chi$ = thermal conductivity, $C_{pi}$ = constant pressure specific heat of component $i$, $M_i$ = molecular weight of component $i$, $h_i^0$ = specific enthalpy of component $i$, $T_i$ = reference temperature, $R$ = universal gas constant, $Sc_l$ = turbulent Schmidt number = 1, $Pr_e$ = turbulent Prandtl number = 1, $C_u$ = 0.09, $C_1$ = 1.44, $C_2$ = 1.92, $\sigma_K$ = 1.0, $\sigma_\epsilon$ = 1.3, $C''_l$ = 2.24/$C_1$, $\partial \psi / \partial z$ = derivative normal to the boundary, $f_{BL}$ equals 1 in the boundary layer next to solid walls and equals 0 outside the boundary layer, $f_1 = 1.0$.

### TABLE III. - BOUNDARY AND INITIAL CONDITIONS

<table>
<thead>
<tr>
<th>Location</th>
<th>Equation</th>
<th>Eq. No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Housing surface</td>
<td>$\bar{u} = \bar{v} = 0$</td>
<td>(45)</td>
</tr>
<tr>
<td>Rotor surface and apices</td>
<td>$\bar{u} = \frac{\partial \bar{X}}{\partial t}, \bar{v} = \frac{\partial \bar{Y}}{\partial t}$</td>
<td>(46)</td>
</tr>
<tr>
<td>Housing surface</td>
<td>$\bar{T} = 300$ K</td>
<td>(47)</td>
</tr>
<tr>
<td>Rotor surface and apices</td>
<td>$\bar{T} = 300$ K</td>
<td>(48)</td>
</tr>
<tr>
<td>Housing and rotor surfaces, apices</td>
<td>$\frac{\partial \bar{X}}{\partial z} = 0$</td>
<td>(49)</td>
</tr>
<tr>
<td>Housing and rotor surfaces, apices</td>
<td>$K = \epsilon = 0$</td>
<td>(50)</td>
</tr>
<tr>
<td>Intake port</td>
<td>$P_0 = 1.01 \times 10^5$ N/m$^2$</td>
<td>(51)</td>
</tr>
<tr>
<td>Intake port</td>
<td>$T_0 = 300$ K</td>
<td>(52)</td>
</tr>
<tr>
<td>Intake port</td>
<td>$\bar{X}_A = 1.0$</td>
<td>(53)</td>
</tr>
<tr>
<td>Intake port</td>
<td>$\bar{u} = 0$</td>
<td>(54)</td>
</tr>
<tr>
<td>Intake port</td>
<td>$\bar{T} = \frac{D_{DI}}{2(n_1 - \bar{V})} \left[ \frac{(2P_0 - \bar{P})}{\rho} \right]^{1/2}$</td>
<td>(55)</td>
</tr>
<tr>
<td>Intake port</td>
<td>$K = \left( \frac{n_1}{n_1 - \bar{V}} \right)^2$</td>
<td>(56)</td>
</tr>
<tr>
<td>Intake port</td>
<td>$\epsilon = K^{3/2} / \left( n_2 L_1 \right)$</td>
<td>(57)</td>
</tr>
<tr>
<td>Fuel injector</td>
<td>$T_f = 300$ K</td>
<td>(58)</td>
</tr>
<tr>
<td>Fuel injector</td>
<td>$\bar{V} = \left( V_0 \sin \frac{\theta - \theta_1}{\theta_2 - \theta_1} \right) \sin \alpha$</td>
<td>(59)</td>
</tr>
</tbody>
</table>

SHIH

7
TABLE III. – Concluded.

<table>
<thead>
<tr>
<th>Component</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fuel injector</td>
<td>( \ddot{u} = \dot{v} / \cot \alpha )</td>
</tr>
<tr>
<td>Fuel injector</td>
<td>( \frac{\partial P}{\partial \zeta} = 0 )</td>
</tr>
<tr>
<td>Fuel injector</td>
<td>( K = (n_3 \dot{v})^2 )</td>
</tr>
<tr>
<td>Fuel injector</td>
<td>( \epsilon = K^{3/2} \left( \frac{n_4 L_2}{(n_4 L_2)^2} \right) )</td>
</tr>
<tr>
<td>Exhaust port</td>
<td>( \ddot{V} = c_{DE} \left[ 2 (\ddot{P} - P_{exh}) / \rho \right]^{1/2} )</td>
</tr>
<tr>
<td>Exhaust port</td>
<td>( \ddot{u} = 0 )</td>
</tr>
<tr>
<td>Everywhere inside</td>
<td>( \frac{\partial}{\partial y} \dddot{x} = 0 )</td>
</tr>
</tbody>
</table>

In Eqs. (73) to (78), \( \Omega \) is the angular speed of the shaft (\( \Omega = 5000 \text{ rpm} \)). \( \theta_0 \) is the crank angle at which we begin computations (\( \theta_0 = 0 \)). \( \theta_f \) is the crank angle after which the time-step size becomes a constant (\( \theta_f = 1 \text{ radian} \)). The parameter \( n_s \) denotes the time level when \( \theta = \theta_f \) (\( n_s = 800 \)). \( \delta \theta_1 \) is the crank angle traversed by the shaft during the first time step (\( \Delta \theta_1 = 0.001 \delta \theta_2 \)). \( \Delta \theta_f \) is the crank angle traversed by the shaft during each time step after time level \( n = n_s \) or crank angle \( \theta_f \). The parameter \( n_t \) is the total number of time steps needed to traverse \( \delta \theta \) crank angles (\( n_t = 10000 \)).

For computational efficiency, the number of grid points (intersections of grid lines) used should be kept to the minimum required to resolve spatially all significant features of
were placed near the rotor housing, and seals. These finite-difference equations provided the solutions at every grid point and time level presented in the next section.

RESULTS

Numerical solutions were generated to investigate the flow patterns inside the combustion chambers of the rotary engine depicted in Fig. 1 during intake, compression, gaseous fuel injection, expansion, and exhaust. Density, mass fractions of fuel and air, x- and y-components of the velocity, energy, temperature, pressure, turbulent kinetic energy, and dissipation rate of turbulent kinetic energy were calculated at 1071 (i.e., 51-by-21) grid points (see Fig. 3) for 10,000 time levels. Only results for the velocities (velocities relative to the stationary housing) are presented. These results are shown in Figs. 4 to 16 illustrating the flow patterns. Do not infer velocity gradients from these figures because the velocity vectors were plotted to best illustrate the flow patterns.

Figures 4 to 6 illustrate the flow patterns during the intake process. During the early part of the intake process, both the intake and exhaust ports were open. Accordingly, air was found to enter the combustion chamber through the intake port and to exit through the exhaust port as shown in Fig. 4. Here, the stagnation pressure of the air entering the combustion chamber $P_0$ was made slightly higher than the exhaust port back pressure $P_{exh}$ (i.e., slightly turbocharged) so that back flow through the intake port did not occur. Later in the intake process, only the intake port was open and a high speed jet issued into

SHIH
TABLE IV. - COORDINATE TRANSFORMATION

Equations

\[
X(\xi, \eta, \tau) = X_2(\xi, \eta)h_1(n') + X_1(\xi, \eta)h_2(n') - K_2(\xi) \frac{\partial Y_2(\xi, \eta)}{\partial \xi} h_3(n')
\]

\[
\frac{\partial Y_1(\xi, \eta)}{\partial \xi} h_4(n') - K_1(\xi) \frac{\partial h_4(\xi, \eta)}{\partial \xi} h_4(n')
\]

(79)

\[
Y(\xi, \eta, \tau) = Y_2(\xi, \eta)h_1(n') + Y_1(\xi, \eta)h_2(n') + K_2(\xi) \frac{\partial X_2(\xi, \eta)}{\partial \xi} h_3(n')
\]

\[
+ K_1(\xi) \frac{\partial h_4(\xi, \eta)}{\partial \xi} h_4(n')
\]

(80)

\[
t = \tau
\]

(81)

where

\[
h_1(n') = 2(n')^3 - 3(n')^2 + 1
\]

(82)

\[
h_2(n') = -2(n')^3 + 3(n')^2
\]

(83)

\[
h_3(n') = (n')^3 - 2(n')^2 + n'
\]

(84)

\[
h_4(n') = (n')^3 - (n')^2
\]

(85)

\[
n' = 0.5 \left( B_\eta + 1 \right) - B_\eta \left\{ 1 + \left[ \frac{B_\eta + 1}{B_\eta - 1} \right] (2n-1) \right\}
\]

(86)

\[
1 - \frac{A_L - A_T}{A_T - A_L} = 0.5 \left( B_\xi + 1 \right) - B_\xi \left\{ 1 + \left[ \frac{B_\xi + 1}{B_\xi - 1} \right] (2\xi-1) \right\}
\]

(87)

\[
1 - \frac{V_L - V_T}{V_T - V_L} = 0.5 \left( B_\xi - 1 \right) + B_\xi \left\{ 1 + \left[ \frac{B_\xi + 1}{B_\xi - 1} \right] (2\xi-1) \right\}
\]

(88)

\[
K_1(\xi) = K_2(\xi) = 2 \left[ X_1(\xi, \eta) - X_2(\xi, \eta) \right]^2 + \left[ Y_1(\xi, \eta) - Y_2(\xi, \eta) \right] \frac{1}{2}
\]

(89)

\[a\]X_1, Y_1, X_2, and Y_2 are given by Eqs. (1) to (12) in Table I with

the parameters A and V related to \( \xi \) as shown by Eqs. (87) and (88) in this

table; B_\xi = 0.1; B_\eta = 0.3.

the combustion chamber (Figs. 5 and 6). In

Fig. 6, a very small region of recirculating

flow can be observed near the intake port.

However, this recirculating flow region was

found to appear aperiodically and then disap-

pear. The disappearance of the recirculating

flow is due to the general motion of the entire

combustion chamber in one direction.

Figures 7 and 8 illustrate the flow pat-

terns during compression. Recirculating flows

were not observed during the compression

process.

Figures 9 to 11 illustrate the flow pat-

terns during gaseous fuel injection. Here

gaseous fuel (octane) was injected into the com-

bustion chamber between crank angles \( \Theta_1 = 7.4 \)

rad or 425\(^\circ\) and \( \Theta_2 = 8.5 \) rad or 487\(^\circ\). The

velocity of the fuel jet was given by Eqs. (59)

and (60) in Table III. The maximum speed of the

fuel jet was \( V_0 = 60 \) m/sec and the fuel was
injected at an angle of $\alpha = 45^\circ$ (see Fig. 1). When the fuel was first injected, its momentum was low in comparison to the momentum of the gas already inside the combustion chamber. Consequently, at this time the fuel jet could not penetrate very deeply into the combustion chamber as shown in Fig. 9. Later as the momentum of the fuel jet increased, it penetrated much deeper into the combustion chamber and a relatively large recirculating flow region can be observed as shown in Fig. 10. Towards the end of the fuel injection process, the momentum of the fuel jet was again low in comparison to the momentum of the gas inside the combustion chamber and the fuel jet was deflected as shown in Fig. 11.
Results for the fuel injection process should be interpreted carefully. First, in realistic rotary engines, a liquid fuel instead of a gaseous fuel is injected. A liquid fuel jet has much more momentum than a gaseous fuel jet at identical jet velocities. Second and more important, in realistic rotary engines, the fuel jet invokes a three-dimensional phenomenon known as a jet in a crossflow. Jets in crossflows cannot be accounted for by two-dimensional numerical studies.

Figures 12 and 13 illustrate the flow patterns during the expansion process. No recirculating flow patterns were observed.

Figures 14 to 16 illustrate the flow patterns during exhaust. Here, we can observe some recirculating flows near the exhaust port as the gas rushed out of the combustion chamber.

The results presented above were not compared with other numerical or experimental data because they do not yet exist. As noted in the Introduction, the present study constitutes the first attempt at numerically studying the unsteady, multidimensional flow inside rotary engine combustion chambers.

CONCLUDING REMARKS

The results generated in this study illustrate some of the major features of the flow patterns taking place inside the combustion chambers of a two-dimensional rotary engine with gaseous fuel injection. From the results obtained so far, it appears that the mathematical model developed here can be used to study the effects of a number of engine design and operating parameters (such as those relating to the fuel injector, rotor pocket, engine speed, and intake-exhaust manifold pressure ratio) on the flow patterns.

Clearly, a two-dimensional study cannot account for all the important factors which influence combustion inside a real rotary engine. In particular, the effects of a jet in a crossflow (taking place during intake and fuel injection) as well as Taylor vortices can only be accounted for by a three-dimensional analysis. Since jets in crossflows and Taylor vortices can significantly affect the flowfield inside rotary engines, a three-dimensional numerical study is already underway.

REFERENCES


SHIH

13


A Two-Dimensional Numerical Study of the Flow Inside the Combustion Chambers of a Motored Rotary Engine

T. I-P. Shih, S.L. Yang, and H.J. Schock

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

Prepared for the 1986 SAE International Congress and Exposition, Detroit, Michigan, February 24-28, 1986. T. I-P. Shih and S.L. Yang, University of Florida, Gainesville, Florida 32611; H.J. Schock, NASA Lewis Research Center. This work was supported by NASA Grant NAG 3-363.

A numerical study was performed to investigate the unsteady, multidimensional flow inside the combustion chambers of an idealized, two-dimensional, rotary engine under motored conditions. The numerical study was based on the time-dependent, two-dimensional, density-weighted, ensemble-averaged conservation equations of mass, species, momentum, and total energy valid for two-component ideal gas mixtures. The ensemble-averaged conservation equations were closed by a $K-\varepsilon$ model of turbulence. This $K-\varepsilon$ model of turbulence was modified to account for some of the effects of compressibility, streamline curvature, low-Reynolds number, and preferential stress dissipation. Numerical solutions to the conservation equations were obtained by the highly efficient implicit-factored method of Beam and Warming. The grid system needed to obtain solutions were generated by an algebraic grid generation technique based on transfinite interpolation. Results of the numerical study are presented in graphical form illustrating the flow patterns during intake, compression, gaseous fuel injection, expansion, and exhaust.

Rotary engine; Flowfield; Two-dimensional simulation

Unclassified - unlimited
STAR Category 02