TENTH AUSTRALASIAN FLUID MECHANICS
CONFERENCE - UNIVERSITY OF MELBOURNE
11-15 DECEMBER 1989

A VORTEX METHOD FOR SIMULATION OF THE THREE DIMENSIONAL FLOW FIELD IN A SPARK IGNITION ENGINE

S. KUMAR and H.C. WATSON

Department of Mechanical and Manufacturing Engineering
University of Melbourne, Parkville, Vic. 3052
AUSTRALIA

ABSTRACT

A methodology for simulation of three-dimensional flow in the combustion chamber of a spark ignition engine is described. The methodology is based on a higher order vortex method which uses an explicit velocity kernel for the solution of the flow field. A three-dimensional random walk method is used for simulating the diffusion of vorticity. The methodologies used for satisfying the potential and the no-slip boundary conditions are also described.

INTRODUCTION

The processes taking place within the combustion chamber of an engine can be described as a complex interaction of the fluid-dynamic, thermal and reaction kinetic phenomena. A complete understanding of these phenomena and their interaction is crucial to developing engine designs for optimzing engine combustion and hence performance and emission characteristics. The flow field established in the combustion chamber is the result of this complicated interaction and is turbulent, unsteady and 3-dimensional in nature. Therefore validated mathematical modeling of this flow field is needed to enhance our understanding of the engine combustion and to improve the design of modern engines.

Traditional analyses of flow fields in the engine cylinder have been based on some form of finite-difference treatment of the appropriately averaged Navier-Stokes equation. Such averaging also necessitates an adequate set of relations known as a closure model or a turbulence model to correlate the turbulent flow parameters. However, such differencing methods suffer from several drawbacks which include the necessity of introducing turbulence models on heuristic grounds rather than obtaining information about them from the solution, and numerical diffusion which tends to smooth out the local perturbations. Many of these drawbacks are overcome by the Random Vortex Method due to Chorin (1973) which is essentially a grid free method and relies on mimicking the essential features of the flow field. However, it is limited to the 2-dimensional flow field. Generalization to three dimensions is possible by assuming the vorticity field to be represented by a collection of tubes or filaments as reported by Leonard (1980). In all these methods the vorticity is distributed over a finite region, variously known as vortex blob, vortex filament or vortex tube etc. The necessity to introduce these structures arises from the earlier observations that point vortices will allow arbitrarily large velocities.

Recently a complete rigorous theory accounting for the accuracy, non-linear stability, and convergence of vortex methods in 3-dimensions has been developed by Beale and Majda (1982) and Anderson and Greengard(1985). It is possible to avoid the singularity at the point vortices by proper choice of the velocity kernels and these provide higher order accuracy compared to the previous vortex methods. Our simulation methodology is based on the above concept.

MATHEMATICAL MODEL

The continuity and the momentum equations can be written as follows:

$$\frac{D\rho}{Dt}+\rho(\nabla.\vec{U})=0 \qquad (1)$$

$$\rho\frac{D\vec{U}}{Dt}-\rho\nabla f-\nabla P+\mu\nabla^2\vec{U}+\frac{1}{3}\mu\nabla(\nabla.\vec{U})=0 \qquad (2)$$

where ρ is the density, $\vec{U}$ is the velocity vector, f is the energy potential, P is the pressure and μ is the viscosity.

Following assumptions are made:

1. The Mach number of the flow in the engine is quite small, so the compressibility effects can be ignored.
2. The fluid is barotropic i.e. the density is a single valued function of the pressure.

With the use of the above assumptions the model equations can be simplified and written as follows:

$$\nabla.\vec{U}=0 \qquad (3)$$

$$\frac{D\vec{\omega}}{Dt}=(\vec{\omega}.\nabla)\vec{U}+\nu\nabla^2\vec{\omega} \qquad (4)$$

where $\vec{\omega}=\nabla\times\vec{U}$ is the vorticity vector and ν is the kinematic viscosity.

The above two equations can be seen as simultaneous convection and diffusion of vorticity. The solution algorithm adopted is the fractional step method, which simulates the convection and the diffusion processes sequentially rather than simultaneously, with the expectation that the error in so doing will converge to zero as the time step approaches zero. Next we describe the methodology adopted for the solution of each of the two components, namely the convection process and the diffusion process.

CONVECTION

The equation to be solved is the following:

$$\frac{D\vec{\omega}}{Dt}=(\vec{\omega}.\nabla)\vec{U} \tag{5}$$

The Lagrangian algorithm used for the solution is due to Beale and Majda (1985) and is presented below.

$$\vec{U}_i(t)=\sum_j K_\delta(\vec{x}_i(t)-\vec{x}_j(t))\vec{\omega}_j(t)h^3 \tag{6}$$

$$\frac{d}{dt}(\vec{\omega}_i(t))=(\vec{\omega}_i(t).\nabla)\vec{U}(\vec{x}_i(t),t) \tag{7}$$

where $\vec{x}$ is the position vector and h is the initial mesh width and i and j refer to two points in the mesh. K_δ is a smooth kernel and is defined as convolution of the velocity kernel K and a smooth function ψ_δ as follows:

$$K_\delta=K*\psi_\delta \tag{8}$$

The functions ψ and ψ_δ are defined as follows:

$$\psi:R^3\longrightarrow R,\ \psi_\delta(\vec{x})=\frac{1}{\delta^3}\psi(\frac{\vec{x}}{\delta}) \tag{9}$$

where δ is a smoothing parameter. The scalar function ψ satisfies the conditions given by Anderson and Greengard (1985). These conditions decide the accuracy and the stability of the vortex method. The smoothing scalar function ψ_δ is often called the cutoff function, smoothing function, blob function or core function in the literature. It may be noted that the velocity kernel K arises in the vortex method due to the following convolution:

$$\vec{U}(\vec{x},t)=(K*\vec{\omega})(\vec{x},t) \tag{10}$$

The smoothing of the kernel K by the function ψ_δ can be interpreted as the approximation of the vorticity distribution by a sum of blobs of prescribed shape as described by Hald (1979).

Following Beale and Majda (1985) we use fourth order accurate and explicit functions ψ_δ and K_δ as follows:

$$\psi_\delta(r)=\frac{1}{4\pi\delta^3}(-3e^{-r^3/\delta^3}+12e^{-2r^3/\delta^3}) \tag{11}$$

$$K_\delta(r)=-\frac{1}{4\pi r^3}(1+e^{-r^3/\delta^3}-2e^{-2r^3/\delta^3}) \tag{12}$$

where r is the distance between two points. The smoothing parameter δ is chosen as follows:

$$\delta=h^{0.5} \tag{13}$$

Since the velocity kernel given by equation (12) is an explicit kernel, the Laplacian operator in equation (7) can be evaluated explicitly. Thus solution of the convection problem is achieved by solving equations (6) and (7).

DIFFUSION

The equation to be solved is the following:

$$\frac{\partial\vec{\omega}}{\partial t}=\nu\nabla^2\vec{\omega} \tag{14}$$

This equation also must be solved in the Lagrangian framework in order that it is compatible with the convection problem. The solution of this equation is given by the 3-dimensional Green's function as follows:

$$Gr(\vec{x},t)=\frac{1}{(4\pi\nu t)^{3/2}}\exp(-\frac{|x|^2}{4\nu t}) \tag{15}$$

The above equation can be seen as product of three one-dimensional Green functions. Furthermore, consider the following probability density function P of a Gaussian variable η with a zero mean and a standard deviation σ:

$$P(\eta)=\frac{1}{(2\pi\sigma^2)^{1/2}}\exp(-\frac{1}{2\sigma^2}\eta^2) \tag{16}$$

Now, let $\sigma=\sqrt{2\nu t}$. It can be easily seen that the following holds:

$Gr(\vec{x},t)\equiv P(\eta_1,\eta_2,\eta_3)$, where (η_1,η_2,η_3) is a 3-dimensional random vector.

Thus the solution of equation (14) can be achieved by three dimensional displacement of vorticity in three perpendicular directions using a set of three Gaussian random numbers, each having a zero mean and a standard deviation $\sigma=\sqrt{2\nu t}$. This displacement is akin to the Random Walk method of Chorin (1978) and is added to the the convective motion calculated from the solution of the convection problem. Next we describe the boundary conditions which the flow must satisfy.

NO-LEAK BOUNDARY CONDITION

The relative fluid velocity at the boundary must not have a component normal to the wall, as this would imply the fluid is leaking from the surface. This leads to the following Newmann boundary value problem:

$$\nabla\phi.\vec{n}=-\vec{U}_{rel}.\vec{n} \tag{17}$$

where $\vec{U}_{rel}$ is the fluid velocity at the boundary relative to the velocity of the boundary, $\vec{n}$ is the unit outward normal at the boundary, and the potential ϕ is governed by the following equation:

$$\nabla^2\phi=0 \tag{18}$$

The methodology for the solution of the above equation is described in detail by Hess and Smith (1962) and consists of describing ϕ as the potential of a source density distribution $\lambda(p)$ over the surface which leads to the Fredholm integral equation of the second kind over the body surface as follows:

$$2\pi\lambda(p)-\iint_S\frac{\partial}{\partial n}(\frac{1}{r(p,q)})\lambda(q)dS=-\vec{U}_{rel}.\vec{n} \tag{19}$$

where p and q are two points on the body surface S and r is the distance between the two points. The piston, the cylinder and the cylinder head surfaces are approximated by several triangular

elements as shown in figure 1 and this approximation is used to discretize the integral equation to arrive at the following system of linear algebraic equations:

$$\sum_{j}^{N} A_{ij}\lambda_j = -(\vec{U}_{rel}\cdot\vec{n})_i, \quad i=1,N \tag{20}$$

where N is the number of triangular elements, A_{ij} is the normal velocity induced at the control point of the i_{th} element due to the unit source density at the control point of the j_{th} element.

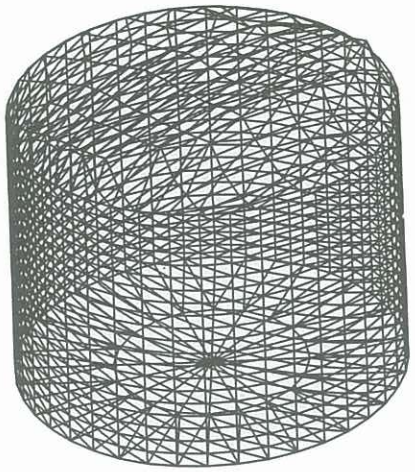

Figure 1 Approximation of the surfaces using triangular elements.

The control point is chosen to be the same as the centroid of the triangular element. The set of equations represented by equation (20) is fairly large depending on the number of elements and thus the surface detail represented. An iterative technique using the conjugate gradient method is employed to solve the set of equations. The algorithm is completely vectorizable and typically for 1600 elements the solution requires about 20 minutes of Cyber 180 model 990 processor time.

NO-SLIP BOUNDARY CONDITION

In a real flow, the velocity of the fluid at solid boundaries must remain the same as the velocity of the boundary. Any slip velocity at the boundary is annihilated and vorticity is produced. Thus the solid boundaries act as a source of vorticity in the flow which then diffuses away from the boundary into the interior flow. Many researchers have used a method similar to that of Chorin (1978), which uses the approximate 2-dimensional boundary layer equations to generate vortex sheets, which then are split up into various elements. In our methodology, we use the Stokes theorem for each of the triangular elements to generate the vorticity at the control point of that element. This is done as follows:

1. Since the no-leak condition is satisfied at the control point of each element, the velocity at that point must be in the plane of the element. Thus the global x-y-z axes are transformed to the local t_1-t_2-n axes, where $\vec{t}_1$ and $\vec{t}_2$ are two perpendicular unit vectors in the plane of the element and $\vec{n}$ is the unit outward normal vector to that element.

2. In the local co-ordinate system, the vorticity can be in the direction of $\vec{n}$ only, in other words, the vorticity cannot have components in the $\vec{t}_1$ and $\vec{t}_2$ directions.

3. It is assumed that the slip velocities in the $\vec{t}_1$ and $\vec{t}_2$ directions, calculated at the control point of each of the elements, are uniform over that element.

4. The generated vorticity is assigned to the control point of that element.

Now the generated vorticity can be calculated from the application of Stokes theorem to the element as follows:

$$\xi_n = \oint \vec{U}_{slip}\cdot d\vec{r} \tag{21}$$

where ξ_n is the vorticity generated per unit area in the direction of the unit outward normal $\vec{n}$ and $\vec{U}_{slip}$ is the slip velocity in the local coordinates. ξ_n is transformed to the global x-y-z coordinate system to obtain the three components of the generated vorticity. This vorticity becomes part of the flow and diffuses away from the surface into the interior flow.

COMPUTATIONAL SCHEME

The computations start at the crank angle corresponding to the closed inlet valve. The initial velocity distribution is assumed known from measurements at this crank angle, at all of the points in the uniform starting mesh shown in figure 2. This mesh is used for placement of

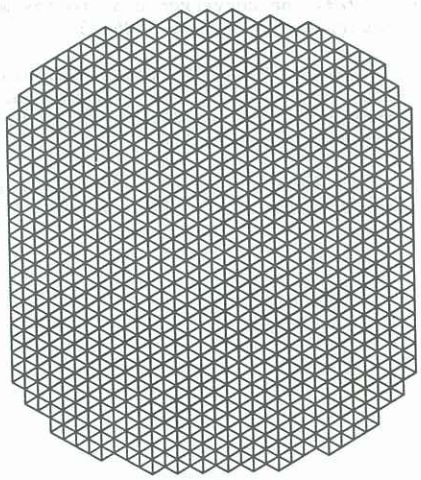

Figure 2 Initial mesh for placement of particles

initial vorticities as well, by differentiating the initial velocity distribution. The computation proceeds in the same sequence as the steps are described in the paper. Each computational module has been developed and tested separately. The programming language used is vector Fortran and

care is taken to vectorize the code to the optimum extent. The code has been developed for Cyber 180 model 990 computer. It is expected that we produce preliminary results of our computations at the paper presentation.

CONCLUSIONS

A higher order vortex method using an explicit velocity kernel is adopted for the simulation of the flow field in an spark ignition engine. The solution of the 3-dimensional Navier-Stokes equation in vorticity form is achieved using a fractional step method, whereby the convection and the diffusion processes are simulated sequentially. The potential boundary condition is satisfied by the solution of the Poisson's equation. The no-slip boundary condition is satisfied by the application of Stoke's theorem, which is used to generate appropriate amount of vorticity at the boundary.

ACKNOWLEDGMENT

We acknowledge the support of Australian Research Council for providing manpower funding. We also acknowledge the University Computing Services for providing grants of free computing resources on the Cyber 990 computer.

REFERENCES

ANDERSON, C. and GREENGARD, C. (1985) On vortex methods. SIAM J. Numer. Anal., 22, pp. 413-440.

BEALE, J.T. and MAJDA, A. (1982) Vortex methods: Convergence in three dimensions. Maths. of Comp., 39, No. 159, pp. 1-27

BEALE, J.T. and MAJDA, A. (1985) High order accurate vortex methods with explicit velocity kernels. J. Comp. Phys., 58, pp. 188-208

CHORIN, A.J. (1973) Numerical studies of slightly viscous flow. J. Fluid Mech., 57, pp. 785-796

CHORIN, A.J. (1978) Vortex sheet approximation of boundary layers. J.Comp. Phys., 27, pp. 428-442

HALD, O. (1979) The convergence of vortex methods. SIAM J. Numer. Anal., 16, pp. 726-755

HESS, J.L. and SMITH, A.M.O. (1962) Calculation of non-lifting potential flow about arbitrary three-dimensional bodies, Report No. E.S. 40622, Douglas Aircraft Company, Inc.

LEONARD, A. (1980) Vortex methods for flow simulation. J. Comp. Phys., 37, pp. 289-335