Space distribution of biodiesel fuel injected into a chamber

Peng Ye

EGEE 520

Introduction:

A Direct Injection (DI) diesel engine is an internal combustion engine which injects the diesel fuel directly into the chamber, where the diesel fuel will be mixed with air and combusts because of the heat generated through the compression of gas in chamber. It is widely used because of its low fuel consumption and free of mixture control. The diesel fuel is injected through a device named injection nozzle fitting usually on the top of a chamber, a high pressure of diesel fuel would push it into the nozzle and create a spray effect at the end of nozzle. As a matter of fact, the spray of diesel fuel is usually accompanied with combustion, and spray combustion includes complex processes such as drop formation, collision, coalescence, secondary breakup, evaporation and interaction with turbulence as reviewed by Faeth (1987) and Lefebre (1989) as well as turbulent mixing and chemical reactions.

For a diesel engine, the combustion and emission characteristics are influenced by fuel atomization, nozzle geometry, injection pressure, shape of inlet port and other factors. In order to improve combustion and emission characteristics, which is to improve the efficiency of combustion and decrease the environmental harmful gas from the exhausting gas such as NOx, CO and particle matters, many researchers have investigated how to improve the air-fuel mixing and receive promising achievements. To improve the air-fuel mixing, understanding of the fuel atomization and spray formation processes is a critical point.

Many researchers have performed both experimental and numerical studies describing the characteristics of diesel spray under high pressure. Dennis (1998), Maruyama (2001), Ishikawa (1996), Farrell (1996) etc have employed the shadowgraph and particle image velocimetry at various chamber conditions under experiments to study the characteristics of diesel spray for the fuel injector. Yeom (2001), Su (1996), Chang (2002), Hohmann (2003), Toshimi (1991) etc have established numerical model of diesel spray and compared it with experimental reports.

This study will use finite elements analysis to simulate the injectino process based on Navier-Stokes approach. It is believe that this complex high pressure injection will create serious instability for the whole system, for which the Peclet and Reynolds number will be used to assess the applicability of Navier-Stokes equation in this injection model.

Table 1 shows the real conditions for this process. However, at the beginning we will use some assumed properties to calculate, the reason for which will be addressed later.

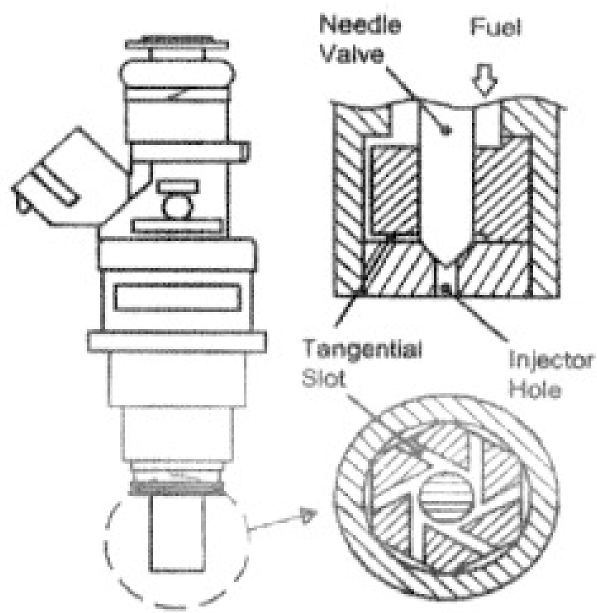

Figure 1: Schematic diagrams Mitsubishi injection nozzle and details of nozzle tip, based on J. Gao, 2005

Table 1: injection condition and fuel properties

Injection duration	1.2ms
Injection pressure	3.6Mpa
Ambient pressure	0.1Mpa
Ambient temperature	300K
Fuel density	880kg/m^3
Fuel surface tension	0.034N/m
Fuel dynamic viscosity	0.00387pa•s

Governing Equation:

For a simply assumed chamber, the diameter of nozzle is 1 cm, and the diameter of chamber is 10 cm. For the symmetry of the system, the lengthwise section will be taken into consideration as a demonstration of this procedure. The fuel is injected at certain pressure into the chamber. To simplify, the pressure from the nozzle is considered to be sustainable during the injection. The whole progress, therefore, is a injected fluid which is continuously derived by its pressure. The liquid is assumed as incompressible.

The governing equations of this model are Navier-Stokes fluid mechanics approach and convection and diffusion shown as follows:

$$\rho \frac{\partial u}{\partial t} + \rho u \bullet \nabla u = \nabla \bullet [-PI + \eta (\nabla u + (\nabla u)^T)] + F$$

with a supplementary equation: $\nabla \bullet u = 0$

$$\delta_{ts} \frac{\partial c}{\partial t} + \nabla \bullet (-D \nabla c) = R - u \bullet \nabla c$$

where ρ denotes the density, t denotes the time, u denote the velocity, P is the pressure, μ is the dynamic viscosity, F is the body force δ_{ts} is the time-scaling coefficient, D is the diffusion coefficient, c is the concentration and R is the reaction rate. The calculation will be based on the coupling of incompressible Navior-Stokes and convection-diffusion transient analysis. Navior-Stokes equation is used to describe the fluid movement, and the diffusion equation is used for computing the fluid movement.

The Level Set Method is used to distinguish the two phases. The basic idea is to denote two universal properties with some function with special properties. The concentration c will be used as a key parameters to distinguish the two phases. In detail, denote the density and viscosity used in Navior-Stokes are:

$$\rho = \rho_a + c * \left(\rho_f - \rho_a \right)$$

$$\mu = \mu_a + c * (\mu_f - \mu_a)$$

define the concentration c as a function of substance, which is as follows:

c=1, when it is biodiesel fuel;

c=0, when it is air;

therefore, density and viscosity will be calculated automatically for different substance. The velocity and viscosity are from Navior-Stokes equation.

A simplified two dimensions model is built to describe this procedure as follows. The diameter is defined as assumption. Diameter of injection nozzle is 1cm, while the diameter of chamber is 10cm:

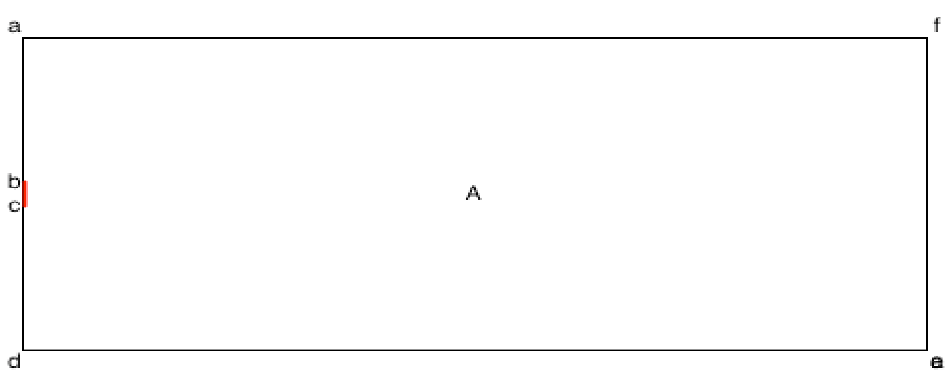

Figure 2: Schematic of model

where, A denotes the nozzle and chamber. The subdomain and boundary conditions for this model is shown in Table 1:

Table 1: Subdomain and boundary conditions for the model

Subdomain	Density (kg/m^3)	Viscosity (pa•s)	Pressure (N/m^2)	Concentration (mol/m^3)
A	ρ	$\mu=\mu_a+c(\mu_f-\mu_a)$	0	0
Boundary				
ab (Wall/no slip)	-	-	-	-
bc (outlet)	ρ	$\mu=\mu_a+c(\mu_f-\mu_a)$	P_o	1
cd (Wall/no slip)	-	-	-	-
de (Wall/no slip)	-	-	-	-
ef (Wall/no slip)	-	-	-	-
fa (Wall/no slip)	-	-	-	-

To simplify the model and shorten the time of calculation, properties are substituted with simple numbers. For ρ_a=1kg/m^3, ρ_f=10kg/m^3, μ_f=1pa•s, μ_a=0.1pa•s, P_o=1000pa. The range of time is 0.3s

Formulating and results:

For this convection and diffusion problem, stability technic of isotropic diffusion in Comsol is employed to stabilize the model. The coefficient of artificial diffusion is 0.5.

Figure 3 & 4 shows the calculated result in different time.

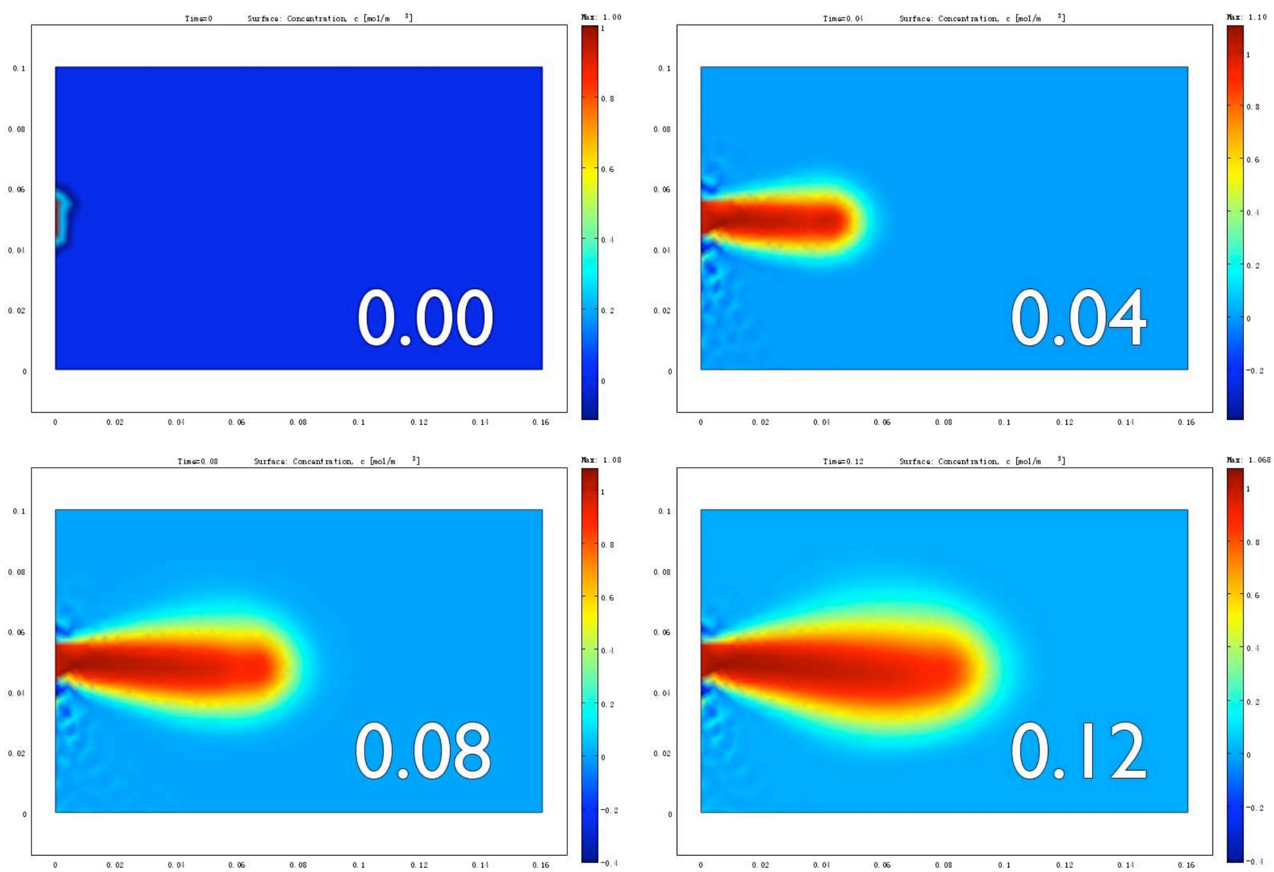

Figure 3: Simulation result

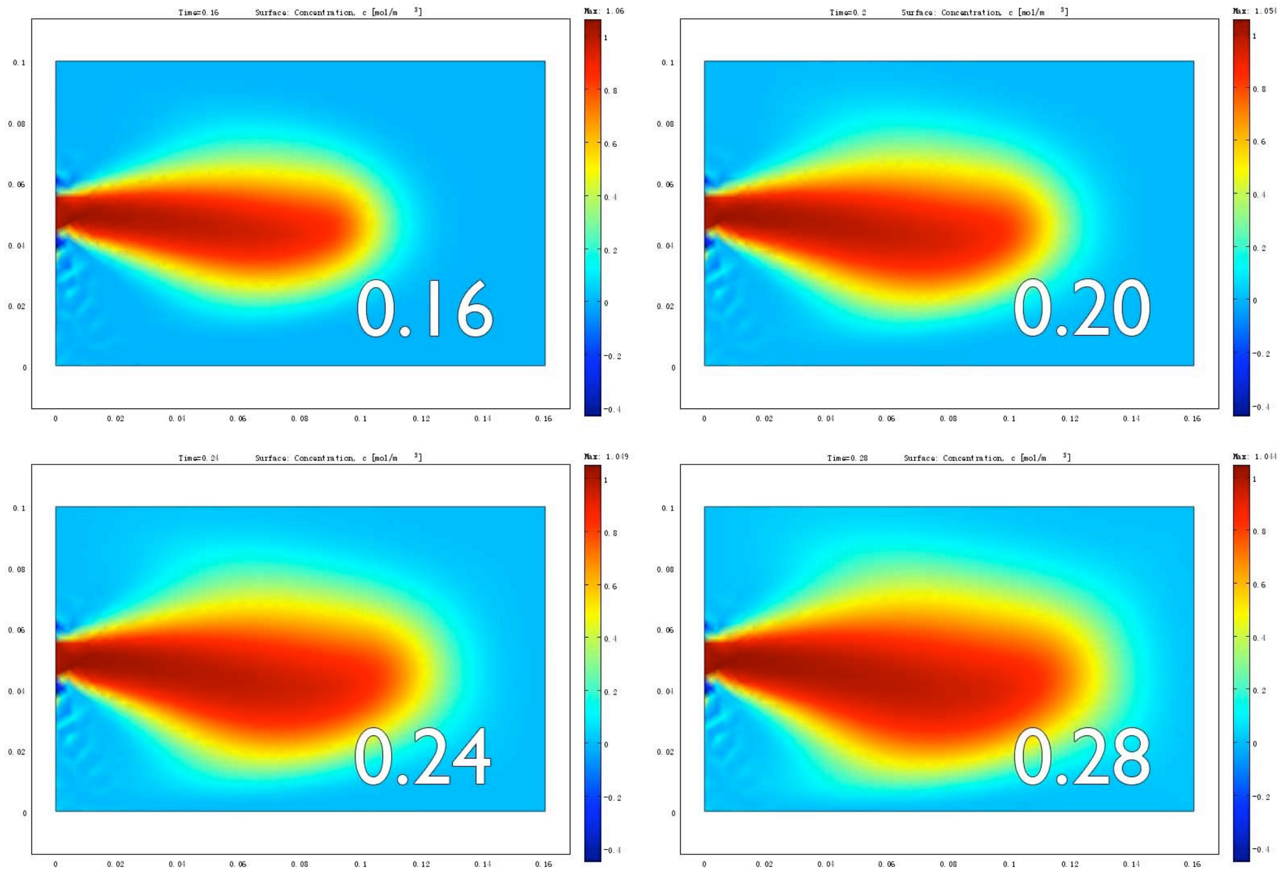

Figure 4: Simulation result continued

Revise and discussion:

During the calculation, I used the assumed properties of model. However, whenthe real properties are implemented into the model, it would not result in expected result rather than error in calculation. The reason for that is based on the limitation of Navier-Stokes equation. The assumptions for using it are as follows:

1. Conservation of mass, momentum and energy.
2. Newton's second law holds.
3. The fluid is a newtonian fluid. The viscosity can be considered as constant and the fluid is isotropic and incompressible.
4. The supplementary equation describes a continuous fluid.

The Assumption 1 and 2 are the basis of classical mechanics, but Assumption 3 is a description of ideal fluid or gases, which will not hold for fluids with complex Reynolds properties. The Reynolds number for a fluid is:

$$Re = \frac{\rho v_s L}{\mu}$$

where Re is the Reynolds number, ρ is the density of fluid, v_s is the velocity of fluid which is derived by the injection pressure, L is the characteristic length which is defined by the mesh volume and μ is the dynamic viscosity.

For the injected fluid, assume the cross section area is *s*, and the length is *l*, therefore the velocity after injection could be calculated through:

$$Psl = \frac{1}{2} \rho sl\, v_s^2$$

therefore:

$$v_s = \sqrt{\frac{2\mathrm{P}}{\rho}}$$

and the Reynolds number will be:

$$Re = \frac{L\sqrt{2\mathrm{P}\rho}}{\mu}$$

For a convection and diffusion problem, the Peclet number is employed to describe the relationship between convective and diffusive term. The Peclet number is defined as:

$$Pe = \frac{L|\beta|}{c}$$

where L is the characteristic length, β is the convective velocity and c is the diffusion coefficient. Peclet number and Reynolds number are measures of the relation between directed convection. Oscillation can occur when the *Pe* or *Re* number exceeds 2.

for high velocity fluid, the convective velocity will easily get large, and consequently the Peclet number would exceed 2. Therefore the solution will become unstable. The stability technic employed in this model is to add a coefficient of artificial diffusion:

$$c_{art} = \frac{L|\beta|}{2}$$

to the diffusion already present in the problem. Then the new Peclet number will be:

$$Pe' = \frac{L|\beta|}{c + c_{art}} = \frac{2\mathrm{L}|\beta|}{2\mathrm{c} + L|\beta|}$$

this Peclet number will approach but never exceed 2, which will stabilize the problem.

However,since L is derived by the mesh unit, which is comparatively constant, the increase in injection pressure as well as density and decrease of dynamic viscosity will result in the increase of the Reynolds number, which means a more complex fluid that Navier-Stokes could handle. A high Reynolds number would also break the fluid, which will not satisfy the continuous requirement.

The characteristic length in our model is 0.01m, therefore the Reynolds number for the assumed properties is:

$$Re=0.01*(2*1000*10)^{1/2}/1=1.414$$

with both the Peclet number and Reynolds number less than 2, oscillation is able to be prevented. At this time, since L is fixed because the size of chamber is fixed, I started with increasing the density a little bit so the Reynolds number exceed 2, say ρ is 100, then the Reynolds number will be:

$$Re=0.01*(2*1000*21)^{1/2}/1=4.47$$

when the density was implemented, the model failed to compute. Now decrease a little bit of the Reynolds number ,say ρ is 50, in this case the Reynolds number is:

$$Re=0.01*(2*1000*50)^{1/2}/1=3.16$$

the model still failed. Now to decrease the Reynolds number further, say the density is 25, then

$$Re=0.01*(2*1000*25)^{1/2}/1=2.23$$

the model failed again. Then to decrease the Reynolds number more further, say the density is 21, then

$$Re=0.01*(2*1000*21)^{1/2}/1=2.05$$

the model was able to compute a result, now increase a little of Reynolds number, say the density is 22, then

$$Re=0.01*(2*1000*22)^{1/2}/1=2.1$$

the model succeeded, then try density is 23, then

$$Re=0.01*(2*1000*23)^{1/2}/1=2.14$$

at this time, the model failed. Therefore we could primarily conclude that the up limit for the Reynolds number of this model is 2.1. To assess this conclusion, the pressure will be used as a variable while other parameter is fixed as the original assumption. The P will be chosen with two number with their Reynolds number are 2.1 and 2.14, therefore:

P1=2205, Re=2.1, the model worked

P2=2289, Re=2.140, the model also worked

As a matter of fact, the model worked until the pressure hit 2500 Pa, which made the Reynolds number:

$$Re=0.01*(2*2500*10)^{1/2}/1=3.38$$

the calculated result is figure 5:

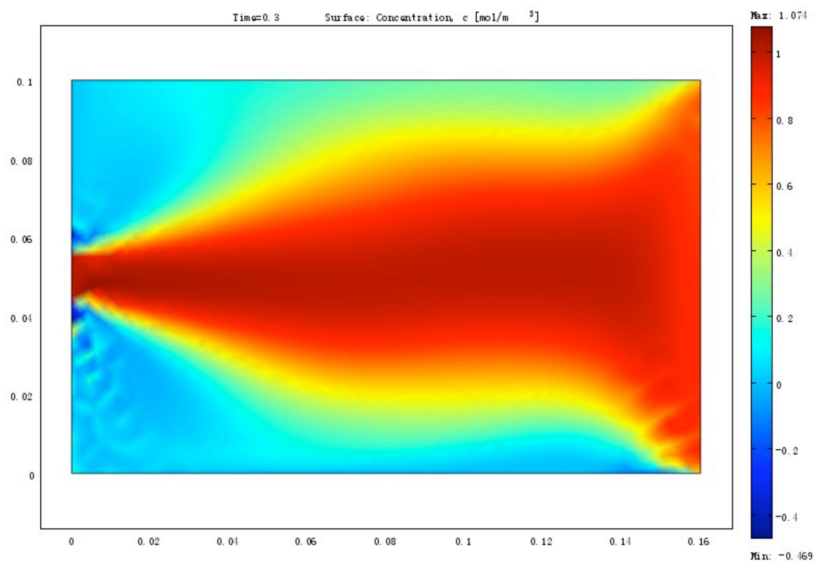

Figure 5: Simulation result when P=2500 Pa

That Reynolds number is different from the Reynolds number derived from density. Now the only left parameter: viscosity is considered through the same method, and it turned out that the model worked until the viscosity hit 0.085 before the oscillation occurred, in which case the Reynolds number:

$$Re=0.01*(2*1000*10)^{1/2}/0.085=16.6$$

The calculated result is figure 6:

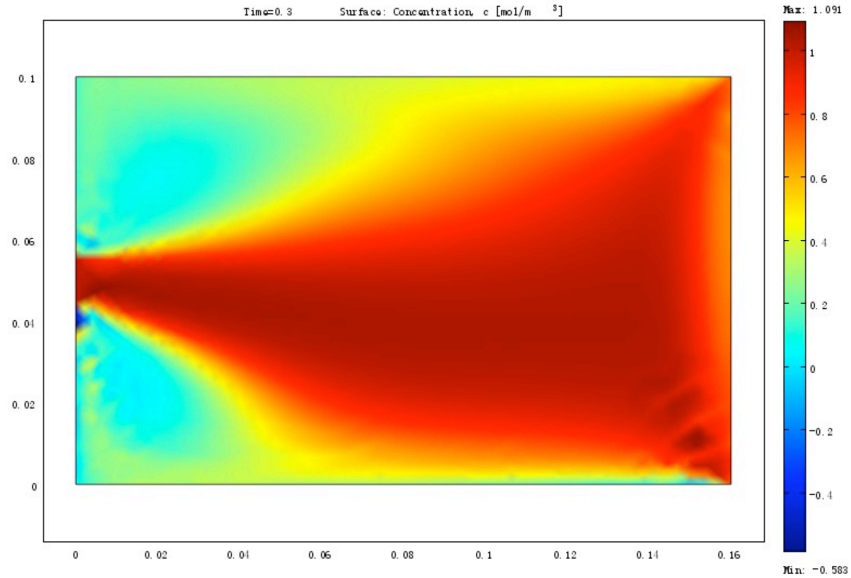

Figure 6: Simulation result when μ=0.085 Pa•s

Which is also different from the other two Reynolds number's up limits. Therefore, the Reynolds number might not be extremely accurate in determining the applicability, but the low variation of them: 2, 3.38 and 16.6 still convincingly demonstrate that Reynolds number could be a good indicator for rough scale.

Obviously from previous calculation high Reynolds number will not fit this model. Now consider real property fluid's Reynolds number:

$$Re=0.01*(2*3600000*880)^{1/2}/0.00387=2.06*10^{5}$$

This Reynolds number is too high for this model to solve. However,a supplement for this study is that the characteristic length L could be modified by fining the element to reduce the Reynolds number. To get a Re of 2 with real properties, the L will be:

$$L=(2*0.00387)^{2}/(2*3600000*880)=9.45*10^{-15}m$$

which is too small for personal computer to calculation, and consequently I could not exam the idea. But it might be worth to compute it if there is comparatively suitable super computer.

Conclusion:

The level set method which couples with Navier-Stokes and convection and diffusion could be used to solve injection problem in two phases. However,this method could only solve those situation without serious oscillation. Peclet and Reynolds number could be employed to indicate that whether the fluid is solvable. Through the implement of stabilization technique of comsol, the Peclet number can be controlled under 2. At the same time, it is found that the solvable fluids have the highest Reynolds number at the range from 2.1 to 16.6. The three parameter of Reynolds number, density, pressure and viscosity have different influences to the solvability of this model, and the density has the highest sensitivity while the viscosity has the lowest. In all, requiring no strict accuracy, the Reynolds number could be used to indicate that whether the oscillation occurs, where the density has the most significant influence in determining the properties of fluid.

Reference:

Dennis LS. SAE Paper, 1998; 980809.

Maruyama Y, Chiba T, Komori M, Saito M, Arai M. Proceedings of ILASS-Asia, 2001; 2001. p. 241 – 6.

Ishikawa N, Niimura K. SAE Paper, 1996; 960770.

Farrell PV, Chang CT, Su TF. SAE Paper, 1996; 960860.

Yeom J, Senda J, Fujimoto H. Proceedings of ILASS-Asia, 2001; 2001. p. 167 – 72.

Su TF, Patterson MA, Reitz RD, Farrell PV. SAE Paper, 1996; 960861.

O'Rourke PJ, Amsden AA. SAE Paper, 1987; 872089.

J. C. Beck, A. P. Watkins, J Comput Phys 182, 586 (2002)

C. A. Felippa, Comput Struct 79, 1313 (2001).

N. Foster, D. Metaxas, Graph Model Im Proc 58, 471 (1996).

J. Gao, D. Jiang, Z. Huang, X. Wang, P I Mech Eng A-j Pow 219, 617 (2005).

J. Gao, D. M. Jiang, Z. H. Huang, Q. Wei, Energ Fuel 19, 1906 (2005).

T. B. Gradinger et al., Combust Flame 124, 422 (2001).

S. Hohmann, U. Renz, Int J Heat Mass Tran 46, 3017 (2003).

C. S. Lee, S. W. Park, Fuel 81, 2417 (2002).

S. P. Lin, R. D. Reitz, Annu Rev Fluid Mech 30, 85 (1998).

G. E. McCreery, C. M. Stoots, Int J Multiphas Flow 22, 431 (1996).

D. W. Stanton, C. J. Rutland, Int J Heat Mass Tran 41, 3037 (1998).

T. TAKAGI, Y. F. CHING, T. KAMIMOTO, T. OKAMOTO, Combust Sci Technol 75, 1 (1991).

S. V. Utyuzhnikov, Flow Turbul Combust 68, 137 (2002).