Experimental and Numerical Analysis of Jet Ignition

Motoi Suetake*, Naomasa Uchida†, and A.Koichi Hayashi‡

Dept. of Mech. Eng., Aoyama Gakuin Univ.
6-16-1, Chitosedai, Setagaya-ku, Tokyo, 157-8572, Japan
*suetake@cow.me.aoyama.ac.jp
†uchida@cow.me.aoyama.ac.jp
‡hayashi@me.aoyama.ac.jp

Abstract

Jet ignition phenomena are experimentally and numerically studied. In the experiment, two types of jet ignition are found out previously and are named as follows this time: Jet Ignition and Auto Ignition. Auto ignition appears in the present study at some experimental conditions: higher equivalence ratio or smaller orifice diameter. Two-dimensional and three-dimensional numerical simulation are also performed solving a unsteady compressive Navier-Stokes equation with a 9 species chemical reaction model. A 2-D simulation reveals jet ignition clearly, but doesn't provide the auto ignition case. A 3-D simulation requires the higher resolution and larger computational size.

Introduction

Jet ignition has been observed in nature and many scientific and engineering applications. Experiments and numerical analyses aimed on prevention of disasters, application for internal combustion engines, and so on, were widely performed[1,2]. Some of the typical examples are seen in flame propagation through an orifice, nozzle, and slit in chambers. Jet ignition can be observed in such a combustion chamber with multiple rooms when a burned gas goes from a driver chamber into a receiver chamber through an orifice and provides a strong ignition, immediate pressure build-up, and violent deflagration. The deflagration sometimes transfers to detonation(DDT). Generally, jet ignition phenomena depend on the following parameters: initial pressure, initial fuel concentration, and chamber geometry; orifice diameter or chamber volume[3].

In the present study a new type of jet ignition occurs in certain conditions. The previously found new type of jet ignition[2] is named as "Auto Ignition" and the characteristic difference between jet ignition and auto ignition in various experimental conditions are studied as shown in Fig.1. Besides, using unsteady two-dimensional Navier-Stokes equations and a full chemistry model, numerical simulation is performed and compared with the experimental results. The three-dimensional numerical simulation is also performed to compare with the experiments and 2-D numerical results.

Experimental Details

The schematic diagram of combustion chamber is shown in Fig. 2. It consist of two rooms which is partitioned by a wall with an orifice. There are two types of rooms. One has the same sizes chamber and the other has different sizes; the large driver and small receiver chambers. The combustion chamber has two optical windows made of quartz for a Schlieren visualization system. Pressure is measured by pressure transducers at the wall of each chamber. A commercial automobile plug is used as an igniter at the bottom center of driver chamber.. Experiments are carried out for the premixed hydrogen/air mixture with equivalence ratio from 0.250 to 0.356 and at the orifice diameter 5, 8, 10, and 12 mm. The process is held at the atmospheric pressure and room temperature. These conditions are kept consistent for the different initial concentrations and orifice diameter. A typical Schlieren system is set to visualize jet ignition phenomena using a Kodak high speed video camera for recording.

Numerical Simulation

We performed the 2-D axisymmetric and 3-D simulations of the flow field in the experimental chamber. The chemicaly reacting fluid was described by Navier-Stokes equations. The air/hydrogen chemical reactions model included 19 forward and backward reactions [4] of 9 species. The N-S equations are solved using Harten-Yee non-MUSCL modified-flux type TVD-upwind scheme for the convective terms, Crank-Nicholson semi-implicit scheme for species production terms, and the second-order central difference method for the viscous and axisymmetric terms. The averaged state on a cell boundary is given by the generalized Roe's average to evaluate the numerical flux in the convective terms.. Strange type-fractional step method is used for time integration in order to keep second order. The 2-D's computational grid is quantitatively the same as the experimental one and its scale is 1/4 using a Cartesian grid for radial direction (dr=0.125 mm) and for vertical direction (dz=0.25 mm). The 3-D grid sizes are 0.1 mm for all direction. In order to adjust the complicated combustion chamber shape, a zonal method with the application of fortified algorithm is used for the present case.

The boundary conditions on the walls are given by the adiabatic, non-slip, and non catalytic condition both for 2- and 3-D simulation and those on the center line are also given by the mirror reflection rule for the 2-D case. The initial conditions at the temperature and pressure are the same as the experiment. The equivalence ratio is 1.0 and orifice diameter is 5 mm for the 2-D simulation. The 3-D simulatiom is performed at equivalence ratio of 0.404.

Results and Discussions

Table 1 shows the types of ignition for the 32 cases depending on the hydrogen concentration and orifice size. Auto ignition is observed at the higher hydrogen concentrations in any orifice condition. This suggests that auto ignition occurs at a high hydrogen concentration. The larger the orifice diameter is, the more auto ignition appears except for the orifice diameter of 10 mm or 8 mm. Schlieren records for both types of ignition are shown in Figs. 3 and 4. From these pictures, auto ignition occurs mostly at the corner of the receiver chamber probably due to the interaction between compression waves and turbulent flow together with a focusing effect at the wall. However, auto ignition is also observed near a usual wall.

The pressure histories in experiments and 2-D numerical simulations are shown in Fig. 5. The 2-D simulation shows a rapid pressure build-up in the receiver chamber as is observed in the experiments. In Fig. 5, the results show that jet ignition phenomena is simulated numerically. The 2-D numerical gas temperature distributions are shown in Fig. 6 that a burned jet gas goes through orifice and into the receiver chamber. This is a case of jet ignition, but is not the case of auto ignition. There are several reasons why auto ignition can not be simulated numerically; a coarse grid size, no turbulence model, and so on. Iso-surfaces of the temperature at 1200 K of the 3-D numerical simulation are shown in Fig. 7. Flame jet goes through an orifice and propagates to the upper side of receiver chamber, but rapidly pressure build-up doesn't occur because the computational space is too small to simulate jet ignition phenomena. Unfortunately, none of the simulations lead to the auto ignition.

Conclusions

Hydrogen jet ignition is studied experimentally using the schlieren technique with a high speed video camera and numerically simulated solving the two- and three-dimensional compressive Navier-Stokes equations with 9 species equations. The experimental results show that jet igntion and auto ignition are observed dependently on the initial conditions, and orifice diameter and hydrogen concentration play an important role on the partition between jet igntion and auto ignition. The results of numerical simulation show a typical jet ignition clearly, but doesn't provide auto ignition this time.

References

[1] Charuel P and Leyer JC, Ninth International Colloqium on the Dynamic of Explosions and Reactive Systems,1983.

[2] Hayashi,A.K. et al, Paper 2661,Proc.of 21st International Symposium on Shock Waves,Great Keppel Island,Australia,july 20-25,1997.

[3] Kanzleiter,T.F.,17th Water Reactor Safety Information Meeting,Rockville,MD,Oct.1989.

[4] Hishida,M.,Hayashi,A.K.,14th International Colloqium on the Dynamic of Explosions and Reactive Systems,1993.

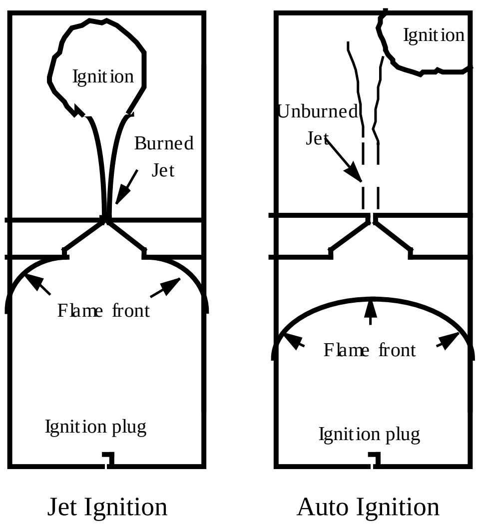

Fig.1 Difference between
Jet Ignition and Auto Ignition.

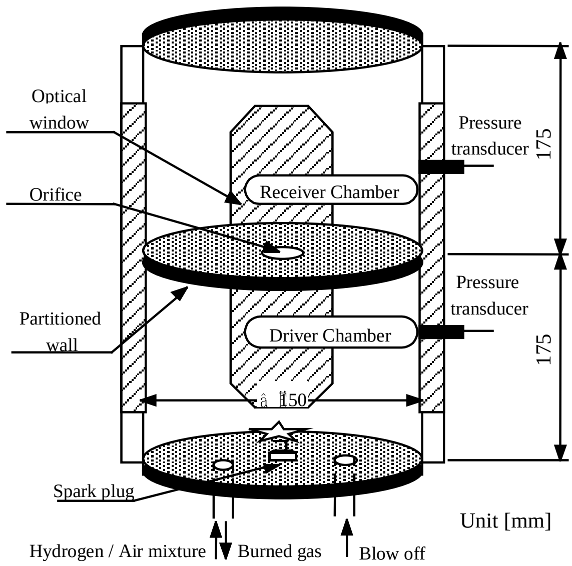

Fig.2 Schematic diagram of
combustion chamber.

Table 1. Combustion types of experimental results.

<table>
<tr><th rowspan="10">Equivalence ratio</th><th colspan="5">Orifice diameter [mm]</th></tr>
<tr><th></th><th>5</th><th>8</th><th>10</th><th>12</th></tr>
<tr><td>0.250</td><td>Jet ignition</td><td rowspan="3">Jet ignition</td><td>Jet ignition</td><td rowspan="4">Jet ignition</td></tr>
<tr><td>0.264</td><td>Unstable</td><td>Unstable</td></tr>
<tr><td>0.279</td><td rowspan="6">Auto ignition</td><td rowspan="6">Auto ignition</td></tr>
<tr><td>0.294</td><td>Unstable</td></tr>
<tr><td>0.309</td><td rowspan="4">Auto ignition</td><td rowspan="2">Unstable</td></tr>
<tr><td>0.325</td></tr>
<tr><td>0.340</td><td rowspan="2">Auto ignition</td></tr>
<tr><td>0.356</td></tr>
</table>

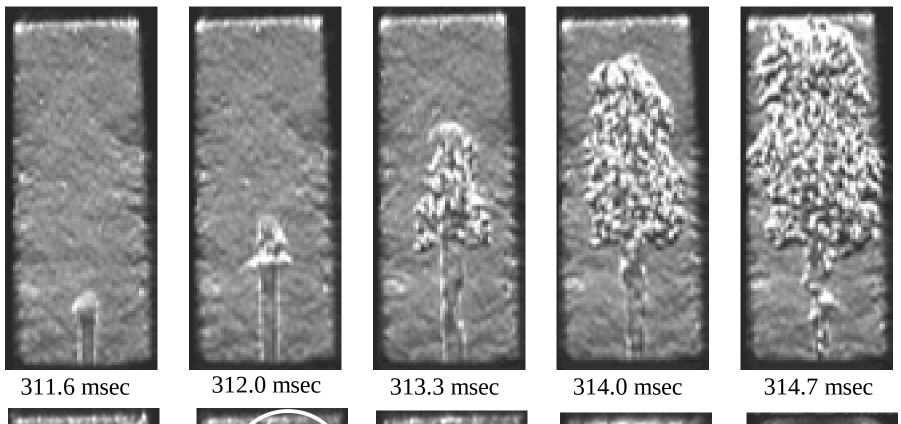

Fig.3 Schlieren record for Jet Ignition. In order to visualize whole receiver chamber, small receiver chamber is used. The orifice diameter is 10 mm andequivalence ratio is 0.250.

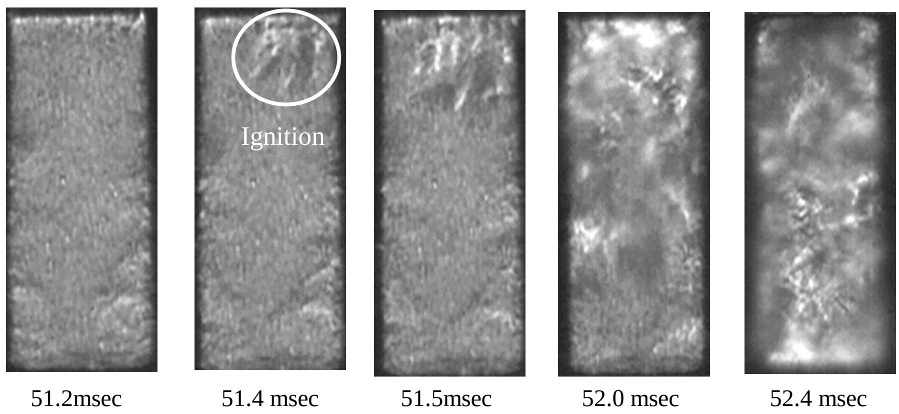

Fig.4 Schlieren record for Auto Ignition. Ignition occurs at the corner of the receiver chamber. The orifice diameter is 10 mm and equivalence ratio is 0.356.

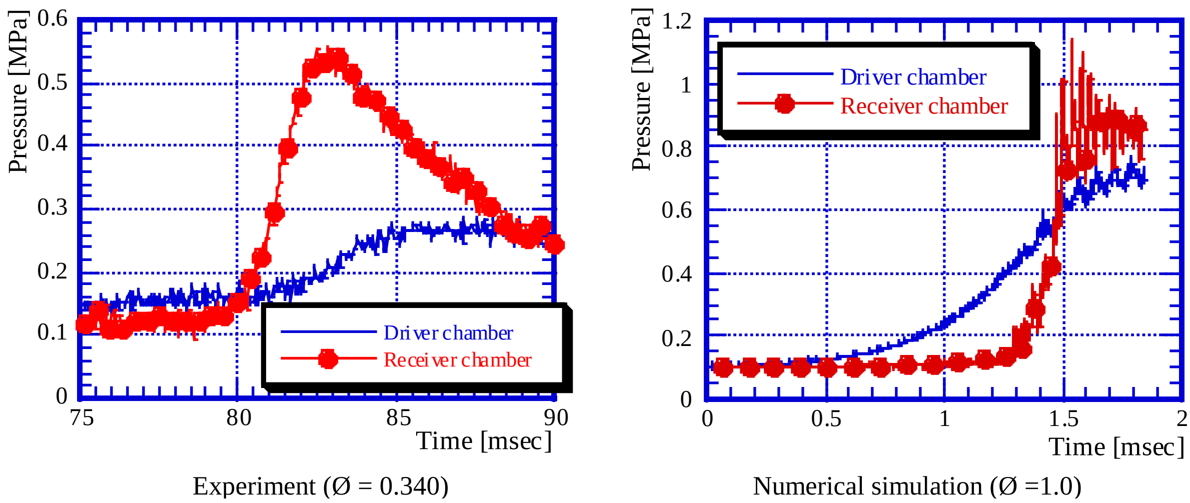

Fig. 5 Comparison of pressure histories between experiment and numerical simulation.

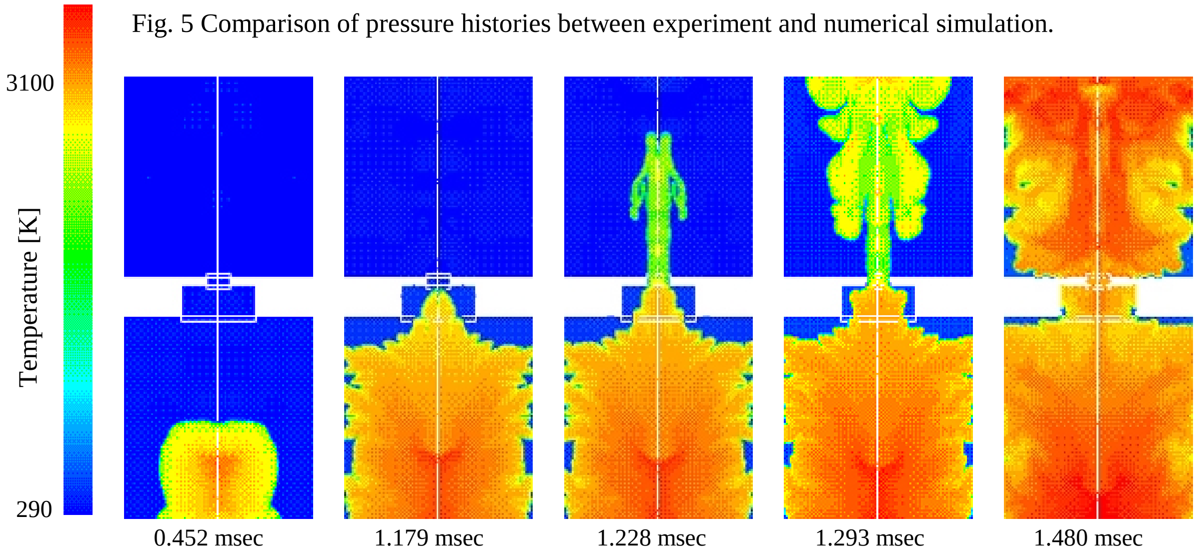

Fig. 6 Temperature distributions of 2-D numerical simulation.

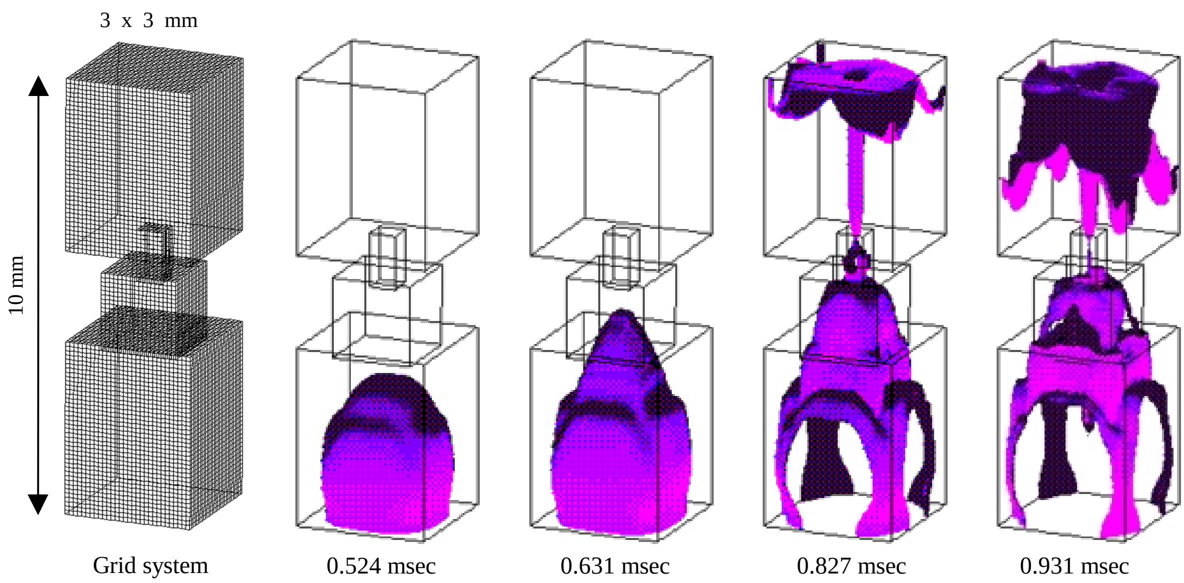

Fig. 7 Grid system and Temperature iso-surfaces of 3-D numerical simulation.