Parallel Modeling of Three-Dimensional Scramjet Combustor and Comparisons with Experiment's Results

Zheng Zhong-hua   Le Jia-ling

(China Aerodynamics Research & Development Center, Mianyang 621000, China)

Abstract

In this paper a parallel simulation of an experimental dual-mode scramjet combustor configuration is presented. Turbulence is modeled with the κ-ε two-equation turbulence model and a 7-species, 8-equation kinetics model is used to model hydrogen/air combustion. The conservation form of the Navier-Stokes equations with finite-rate chemistry reactions is solved using a diagonal implicit finite-volume method. Using about 3,120,000 grid points, the three-dimension flow-fields with equivalence ratio ϕ=0.0 and 0.35 have been respectively simulated on the parallel computer system, obtaining more detailed flow properties than the experiment's results. Wall pressure comparisons between CFD and experiment show fair agreement. For ϕ=0.35, the fuel-penetrating height of the seven injectors are different because of the effects of the boundary layer and the shock wave in the combustor. According to numerical results, if adjusting the locations of the injectors, the combustion efficiency could be improved.

Introduction

Supersonic combustion ramjet engine (Scramjet), in which the combustion process can be supersonic, is a new type of air-breathing propulsion device. It has remarkable performance, great potential superiority, and a good prospect in the field of military and civil aviation in the future. However, the experimental study of scramjet engine puts higher demands on simulating capability of ground test facilities, and greatly increases the test cost and period. Since the flow residence time within a combustor is very short, on the order of one millisecond, it is difficult to measure the flow-fields, generally only get the wall pressure and heat flux. Even using advanced measure technology(Planar Laser Induced Fluorescence), which is expensive, it's still difficult to make three-dimensional measurements, hence difficult to obtain detailed flow properties. With the development of computer technology and the advancement of numerical calculation, Computational Fluid Dynamics (CFD) can provide detailed flow-field characteristics than experimental results and has emerged as an extremely valuable and cost-effective engineering tool in combustor's design and analysis.

Previous work in the area of combustor analysis and predictive capability includes the early work of Billing[1], Billing et al[2], and Waltrup and Billing[3,4]. They developed correlations based on one-dimensional and two-dimensional flow analysis which included such drivers as incoming boundary-layer thickness and maximum combustor pressure rise. These correlations perform well on simple model geometries (axi-symmetric or two-dimensional) but are not generally suitable for more complex three-dimensional problems except for providing directions and trends. Recent computational studies of the three-dimensional combustors are described in papers by Rodriguez et al[5] and Riggins[6]. The computational domains include the use of jet-to-jet symmetry and entire half-duct modeling. Calculations in each case are typically conducted on fixed structured grids of

Report Documentation Page

Report Date	Report Type	Dates Covered (from... to)
23 Aug 2002	N/A	-

Title and Subtitle Parallel Modeling of Three-Dimensional Scramjet Combustor and Comparisons With Experiment's Results	**Contract Number**
	Grant Number
	Program Element Number
Author(s)	**Project Number**
	Task Number
	Work Unit Number
Performing Organization Name(s) and Address(es) Institute of Theoretical and Applied Mechanics Institutskaya 4/1 Novosibirsk 530090 Russia	**Performing Organization Report Number**
Sponsoring/Monitoring Agency Name(s) and Address(es) EOARD PSC 802 Box 14 FPO 09499-0014	**Sponsor/Monitor's Acronym(s)**
	Sponsor/Monitor's Report Number(s)

Distribution/Availability Statement
Approved for public release, distribution unlimited

Supplementary Notes
See also ADM001433, Conference held International Conference on Methods of Aerophysical Research (11th) Held in Novosibirsk, Russia on 1-7 Jul 2002

Abstract

Subject Terms

Report Classification unclassified	**Classification of this page** unclassified
Classification of Abstract unclassified	**Limitation of Abstract** UU

Number of Pages
8

less than 2,500,000 points in multi-zone domains. Among all the numerical simulations the fuel-penetrating heights of different injectors have not been obtained.

This paper is intended to provide a parallel simulation of an experimental dual-mode scramjet combustor configuration. The full duct for the isolator-combustor is then modeled in order to study the fuel-penetrating height of the seven injectors. The grid used for the present calculations is 211×41×121 for the isolator and 211×81×121 for the combustor，about 3,120,000 grid points. Using the Navier-Stokes equations with the addition of conservation of chemical species, the three-dimension flow-fields with equivalence ratio ϕ=0.0 and 0.35 have been respectively simulated on the parallel computer system，obtaining more detailed flow properties than the experiment's results. Wall pressure comparisons between CFD and experiment show fair agreement.

Governing Equations and Numerical Method

In general curvilinear coordinate, the full, three-dimensional Navier-Stokes equations coupled with chemical nonequilibrium processes in a nondimensionalized conservation form are written as

$$\frac{\partial U}{\partial t}+\frac{\partial E}{\partial \xi}+\frac{\partial F}{\partial \eta}+\frac{\partial G}{\partial \zeta}=\frac{1}{R_e}\left(\frac{\partial E_v}{\partial \xi}+\frac{\partial F_v}{\partial \eta}+\frac{\partial G_v}{\partial \zeta}\right)+S$$

where U are the conservation variables, E、F and G are the convective fluxes, Ev、Fv and Gv are the viscous fluxes, and S are the production and reduction rates for the nonequilibrium processes.

The conservative form of the equations is solved using a diagonal implicit finite-volume method, which approximately solves this system using two sweeps of a point Gauss-Seidel relaxation. In the explicit part, the inviscid fluxes are computed using Steger-Warming scheme with a 3rd-order accurate MUSCL interpolation. The viscous fluxes are evaluated using standard central differences. For the implicit calculation, the code uses a diagonal algorithm, which eliminates the expense of inverting large block matrices that arise in chemically reacting flows.

The first step:

$$\left[\mathbf{1}+\Delta\mathbf{t}\left(\lambda_{A\max}+\lambda_{B\max}+\lambda_{C\max}\right)-\Delta tD\right]\overline{\delta U^{n+1}}-\Delta t$$
$$\left(A_{i-1}^{+}\delta U_{i-1}^{n}+B_{j-1}^{+}\delta U_{j-1}^{n}+C_{k-1}^{+}\delta U_{k-1}^{n}\right)=\Delta tRHS$$

The second step:

$$\left[\mathbf{1}+\Delta\mathbf{t}\left(\lambda_{A\max}+\lambda_{B\max}+\lambda_{C\max}\right)\right]\delta U^{n+1}+\Delta t$$
$$\left(A_{i+1}^{-}\delta U_{i+1}^{n}+B_{j+1}^{-}\delta U_{j+1}^{n}+C_{k+1}^{-}\delta U_{k+1}^{n}\right)=\left[\mathbf{1}+\Delta\mathbf{t}\left(\lambda_{A\max}+\lambda_{B\max}+\lambda_{C\max}\right)\right]\overline{\delta U^{n+1}}$$

The Chemical Reacting Model

A combustion model for hydrogen/air[7] mixtures consisting of 8 reactions and 7 species, including 6 reacting species H_2、H_2O、O、H、OH、O_2 and an inert specie N_2, is selected for our computations. The eight reactions are:

1) $H+O_2 \Leftrightarrow OH+O$

2) $O+H_2 \Leftrightarrow OH+H$
3) $H_2+OH \Leftrightarrow H+H_2O$
4) $2OH \Leftrightarrow O+H_2O$
5) $H_2+M_1 \Leftrightarrow 2H+M_1$
6) $H_2O+M_2 \Leftrightarrow OH+H+M_2$
7) $OH+M_3 \Leftrightarrow O+H+M_3$
8) $O_2+M_4 \Leftrightarrow 2O+M_4$

The reaction rates are taken from Evans and Schexnayder[8].

Boundary Condition

The free-stream is supersonic so that all flow variables are known. Exit conditions are set as supersonic extrapolation. The solid wall boundaries are modeled as no-slip, adiabatic, a zero normal gradient of pressure and fully non-catalytic.

Results and Analysis

1. Code Validation

Shock induced combustion phenomena, ranging from decoupled to coupled shock-deflagration systems, were experimentally investigated in the mid 1960's and early 1970's. Calculations of this type of reaction are very demanding in terms of numerical robustness and accuracy, since the reactions usually occur very fast with significant energy release which takes place in a very short distance. In this paper, to validate the code, two results from Lehr's[9] experiment are reproduced numerically. The two cases are

Case	Mixture	$V\infty$(m/s)	$P\infty$(Pa)	$T\infty$(K)
1	H_2/Air	2605	42662	250
2	H_2/Air	1685	42662	250

All cases involve a sphere having a diameter of 15mm moving through a stoichiometric mixture at velocity, pressure and temperature indicated at the above table. Calculations are performed on a $45\times80\times41$ grid.

Experimentally, Lehr found that Case 1 resulted in a coupled shock-deflagration system, whereas Case 2 resulted in a decoupled shock-deflagration system. The temperature contours of Case 1 along with the experimental shock location are shown in figure 1. Case 2 is a condition where a projectile is traveling at a speed lower than the detonation speed of H_2/Air. This case resulted in a decoupled shock-deflagration system shown in figure 2. These two calculations correctly predict the location and the shape of the wave. So the code is right and reliable.

2. Description of the experiment

This section briefly describes the experimental configuration examined in this investigation. The scramjet is essentially a rectangular-section, constant-width duct. On both upper and lower walls there is a backward facing step and, further downstream, a 3° wall expansion only on the upper wall. These two features divide the scramjet into three sections: isolator, combustor and expansion. The steps have a height of 3mm. Injection takes place 95mm downstream of the steps. There are seven injectors on the bottom wall and the diameter of each injector is 1.2mm. Hydrogen is injected through the orifices at sonic conditions，with equivalence ratio $\Phi=0.35$. Other dimensions can be obtained from figure 3. For convenience, the nominal inlet conditions of the vitiated flow are provided in table 1. Table 2 gives the injectant conditions. This experiment has been performed at CARDC.

Table 1: Nominal inlet conditions.

M	2.05	$\mathbf{P_{H2O}}$	0.258
P	0.341Mpa	$\mathbf{P_{O2}}$	0.21
T	1172K	$\mathbf{P_{N2}}$	0.532
U	1417(m/s)		

Table 2: Injectant conditions

M	1.0	$\mathbf{P_{H2}}$	1.0
Pt	2.3Mpa	**D**	1.2mm
Tt	300K	**Φ**	0.35

3. Results and Analysis

1) Φ＝0.0

Without injection the wall pressures at the bottom on the center-plane are compared in figure 4 with the data of CARDC's experiment. The results show fair agreement.

2) Φ＝0.35

For Φ＝0.35，figure 5 shows the wall pressure comparisons at the bottom on the center-plane. The peak pressure is almost equal and the results show also good agreement. Figure 6 gives the center-plane mach number floods. For this low-Φ condition the whole core flow is supersonic. Figure 7 shows the wall pressure comparisons of the numerical simulations between Φ＝0.0 and Φ＝0.35. For Φ＝0.35, there is no upstream interaction and the combustion reactions only affect the downstream flow-field. In order to study the fuel-penetrating height of the seven orifices, the hydrogen-specie flood，the temperature flood and the pressure flood of the cross-section at the center location of the orifices are shown in figure 8. The hydrogen-specie flood shows the fuel-penetrating heights of the seven injectors are different. The temperature flood indicates that the fuel-penetrating height of the two side orifices is low for the impact of the side boundary-layer. At the same time, because of the suppressing of the shock plane, the penetrating heights of the three middle orifices are also low. According to numerical results，if adjusting the locations of the injectors，the combustion efficiency could be improved

.

Conclusions

1. Wall pressure comparisons between CFD and experiment show fair agreement. So the code can be used for studying the data of experiments and getting more detailed properties of the experimental flow field.
2. Because of the effects of the boundary layer and the shock wave in the combustor，for ϕ=0.35，the fuel-penetrating height is low and the combustion efficiency is low. Adjusting the locations of the injectors could improve the combustion efficiency.
3. For Φ＝0.35, the whole core flow is supersonic without upstream interaction. The combustion reactions only affect the downstream flow-field.

Future Work

1. Unsteady analysis may be used for understanding the physics of fuel air mixing and for combustion instabilities.
2. Consider the turbulence-chemistry interactions.
3. More complex reaction mechanisms for hydrogen/air combustion will be needed, particularly

for hydrocarbon fuel.

4. Increase the convergence rates of the code.

Reference

1 Billing，F.S. and Dugger，G.L.，“The interaction of Shock Waves and Heat Addition in the Design of Supersonic Combustors ”， Proceedings of the 12th Symposium on Combustion，Combustion Institute, Pittsburgh，PA, 1969,pp.1125-1134.

2 Billing，F.S. ，Dugger，G.L.，and Waltrup，P.J.，“Inlet –Combustor Interface Problems in Scramjet Engines”，Proceedings of the 1st International Symposium on Airbreathing Engines，Marseilles，France，June 1972.

3 Waltrup，P.J. and Billing，F.S. ，“ Prediction of Pre-combustion Wall Pressure Distribution in Scramjet Engines”，Journal of Spacecraft and Rockets，Vol.10，No.9，1973，pp.620-622.

4 Waltrup，P.J. and Billing，F.S. ，“ Structure of Shock Waves in Cylindrical Ducts”，AIAA Journal，Vol.11，No.9，1973，pp.1404-1408.

5 Rodriguez，C.G.，White，J.A.，and Riggins，D.W.，“ Three-Dimensional Effects in Modeling of Dual-Mode Scramjets”，AIAA 2000-3704，July 2000.

6 Riggins，D.W.，“The Numerical Investigation of a Dual-Mode Scramjet Combustor”，1998 JANNAF Joint Meetings，December 10, 1998.

7 Yungster，S.，Eberhardt，S.，and Bruckner，A.P.，“Numerical Simulation of Shock-Induced Combustion Generated by High-Speed Projectiles in Detonable Gas Mixtures”, AIAA-89-0673.

8 Evan，J.S.，and Schexnayder，C.J.Jr.，“Influence of Chemical Kinetics and Unmixedness on Burning in Supersonic Hydrogen Flames”，AIAA.J.，Vol.18，No.2，Feb.1980.

9 Lehr，H.F.，“Experiments on Shock-Induced Combustion”，Aeronautica Acta,Vol.17,No.4&5. p589-597,1972.

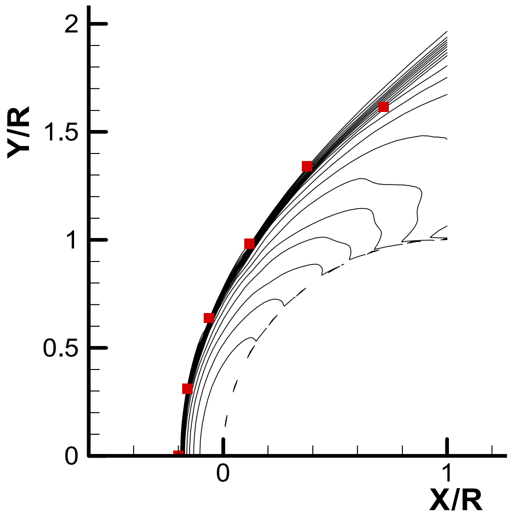


Figure 1: Coupled shock-deflagration system; Temperature contours

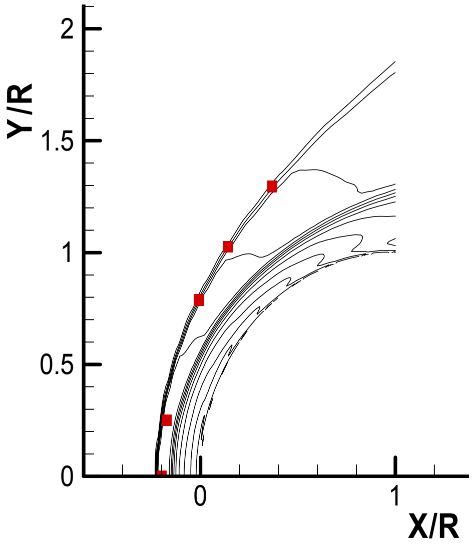


Figure 2: Decoupled shock-deflagration system; Temperature contours

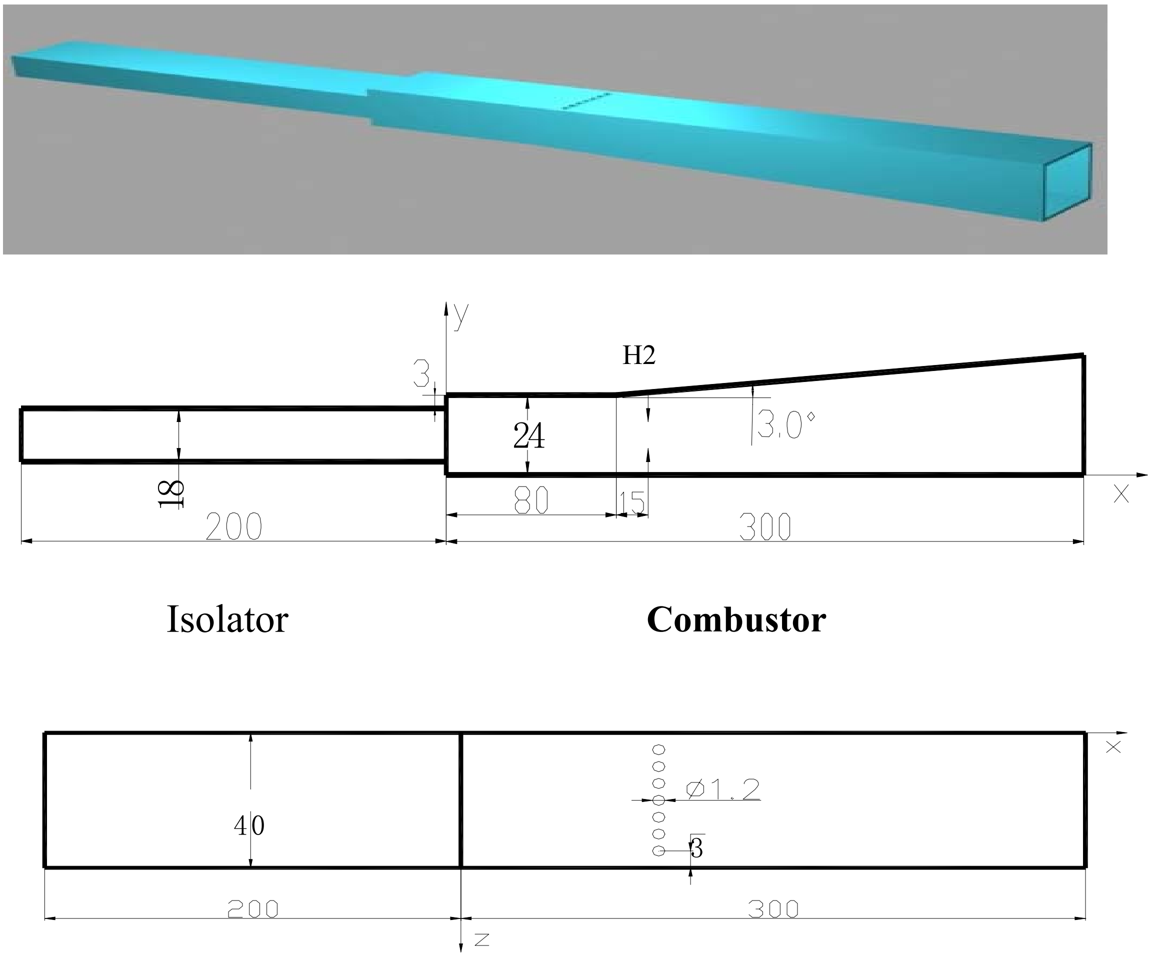


Figure 3: Outline of the dual-mode scramjet experiment

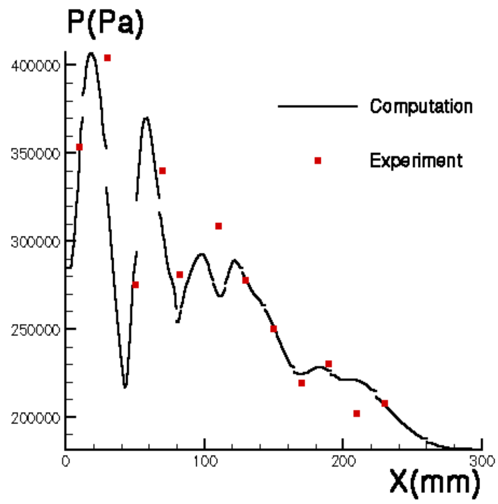


Figure 4: Wall pressure comparisons for Φ＝0. 0

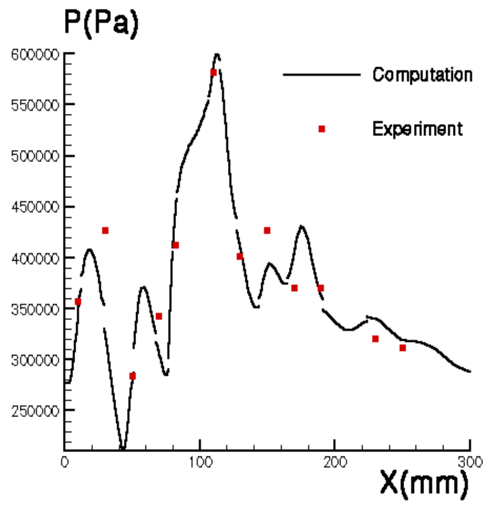


Figure 5: Wall pressure comparisons for Φ＝0. 35

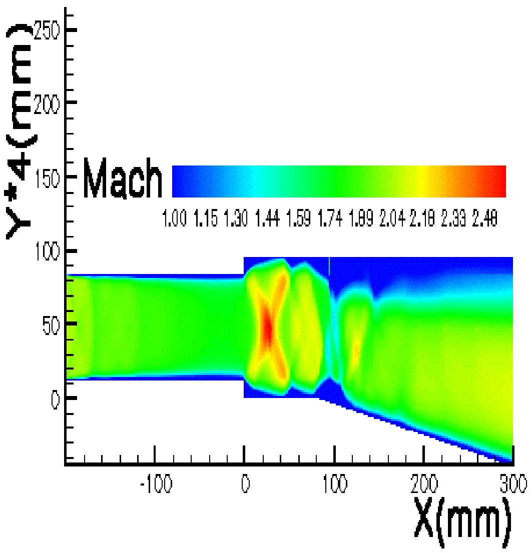


Figure 6: Mach number floods of the cnterplane for Φ＝0.35

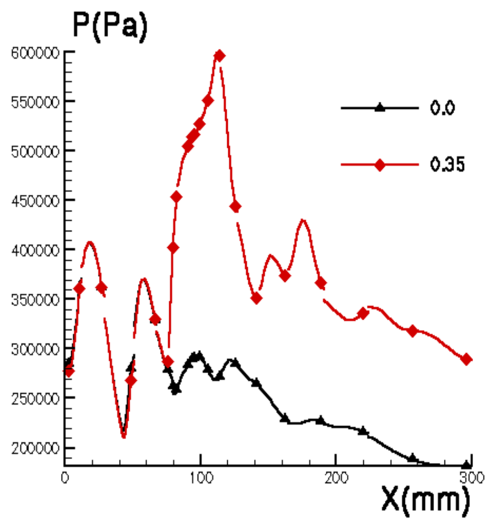
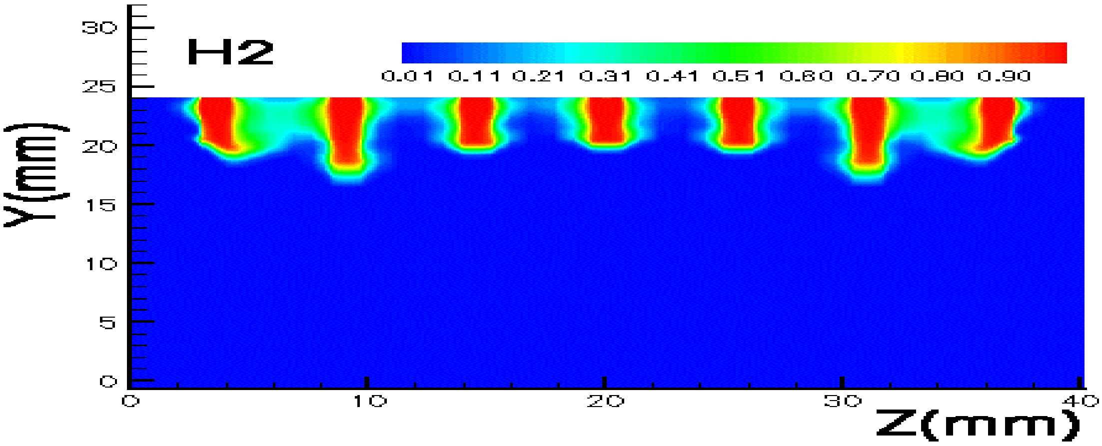
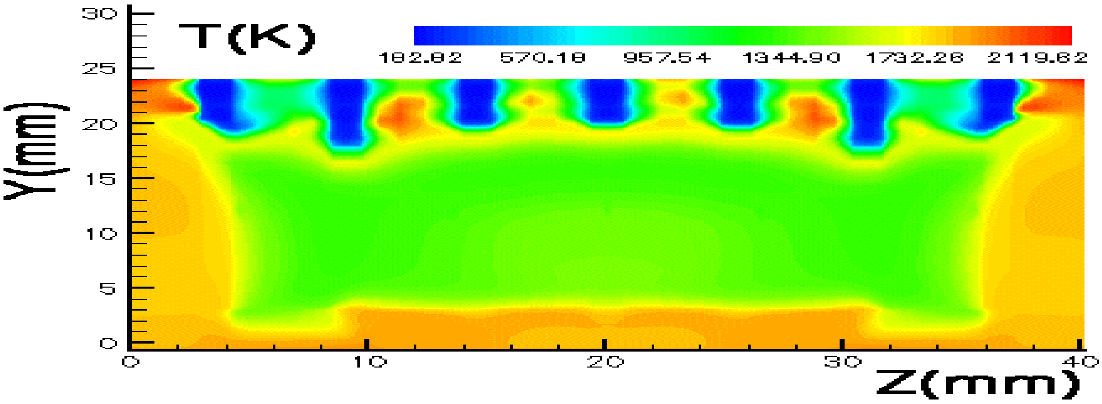


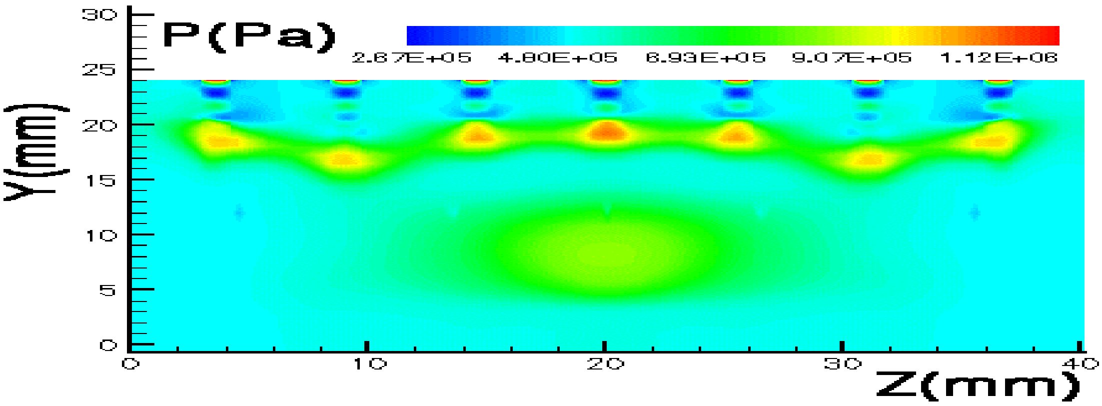
Figure 7:Wall pressure comparisons between Φ＝0.0 and Φ＝0.35



（a）**Hydrogen-specie flood of the cross-section for Φ=0. 35**



（b）**Temperature flood of the cross-section for Φ=0. 35**



(c) Pressure flood of the cross-section for Φ=0. 35

Figure 8: Comparisons of the fuel-penetrating height of seven orifices