NUMERICAL INVESTIGATION OF FUEL MIXING, IGNITION AND FLAME STABILIZATION BY A STRUT INJECTOR IN A SCRAMJET COMBUSTOR

A. Lyubar and T. Sattelmayer

Lehrstuhl für Thermodynamik, TU-München,
D-85747, Garching, Germany

Introduction

The simulation of a supersonic reactive flow is an important tool for the investigation and development of SCramjet engines. Numerical investigations have become less expensive than experiments but in many cases the existing models still do not allow to simulate the complex physical processes with appropriate accuracy. In the simulation of supersonic reactive flows, the development of adequate combustion models are a research field of actual interest. Even the modeling of non reacting supersonic flow with shock waves and complicated shock-boundary interactions is not a trivial task for the CFD-codes available. Such simulations require very fine meshes, which increase the necessary CPU-time remarkably. In addition, the numerical description of turbulent combustion is still in a developing stage. Since the chemical and turbulence time scales are of the same order in the supersonic reacting flows a detailed simulation of the complex chemistry cannot be avoided [1]. The fluctuation of species concentrations and temperature have sufficient influence on the ignition and the flame propagation and must be modeled adequately. Probability density functions (pdf) are usually used to take into account the turbulence-chemistry interaction [2]. Using models with detailed chemistry and pdfs makes the simulation almost impossible for the real 3D geometry because of excessive memory and CPU-time requirements. Hence economical combustion models have to be selected to realize such simulations. In most cases, only highly symmetrical geometries are accessible to the numerical calculations or the real geometry has to by simplified to permit the use symmetry planes, respectively.

For a successful simulation of the reactive flow the accurate solution of the non-reactive flow has to be obtained as a prerequisite. In the past, many proprietary codes with combustion special models have been developed, because the commercially available CFD-codes were not able to simulate supersonic flows. Recently, the capability to simulate such flows was included in some of these codes and using these for research purposes instead of proprietary code became an option. The now available features of local mesh refinement, multigrid methods and parallel computation makes the application of commercially available codes attractive for at least for the simulation of non-reacting flows. On the other hand, efficient combustion models for high-speed combustion are not yet available and have to be developed and implemented as physical submodels. The strategy for the numerical investigation of supersonic flows in the presented research effort is using a commercially available CFD-code for the flow simulation and the implementation of a tailored combustion model into this code.

For the selection of the base code the main criteria were the capability to simulate the supersonic flow precisely and the possibility to implement submodels effectively. As the result of benchmark tests 'Fluent 5' was selected as most suitable for this task. This code has the major advantage of adaptive mesh refinement, that allows the local modification of the mesh during the computation. This option provides a better resolution of the shock waves without significant increase of the number of cells.

In this work the simulation of the non-reacting supersonic flow is presented. The main task of the simulation was to prepare the non-reacting solution for the simulation of reacting flow in the second phase of the project. The flow in a SCramjet with a strut injector [3] was simulated.

Report Documentation Page

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

Title and Subtitle Numerical Investigation of Fuel Mixing, Ignition and Flame Stabilization by A Strut Injector in A Scramjet Combustor	**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

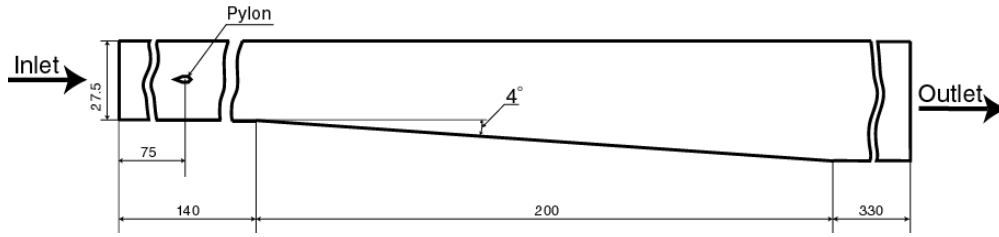| **Report Classification** unclassified	**Classification of this page** unclassified
Classification of Abstract unclassified	**Limitation of Abstract** UU

Number of Pages
5

Figure 1: Geometry of the chamber

The shock system, temperature field and hydrogen distribution are presented subsequently. Preliminary simulations of the reactive flow were realized with the Finite-Rate Model, which is included in 'Fluent 5', in order to investigate potential ignition zones. The model does not yet include the turbulence-chemistry interaction but can, nevertheless, be applied successfully for the initial analysis of the ignition process.

Geometry of the supersonic chamber

The model of the SCramjet propulsion system used for the experimental investigation consists in the preheater, Laval nozzle and the supersonic combustion chamber [3]. The hydrogen preheater and the Laval nozzle are used in the tests to simulate the flight conditions and allow to generate a supersonic flow with a Mach number of 2.15, a total temperature up to 1400 K and a total pressure up to 7.5 bar. The geometry of the chamber is shown in Fig. 1. The chamber has a rectangular cross-section of 25×27.5 mm. The hydrogen injector is installed 75 mm downstream from the inlet of the chamber. The same geometry was used previously with a pylon injector (see Fig. 2 left) being used for the injection of fuel. The pylon is equipped with vortex generators which generate secondary flows and enhance fuel air mixing. The hydrogen is injected through the orifices in the vortex generators which are inclined 30° to the main flow direction. The flow near the pylon has a complex three dimensional behavior. Hence, a simulation of the complete 3D geometry is required. To decrease the computational volume, the geometry of this injector was simplified to obtain a 2D configuration. The numerically most simple configuration would be a strut with a slot injection (see Fig. 2, center) but this geometry is very cumbersome to manufacture. Therefore, the slot was approximated by a row of orifices with the same flow area. This hydrogen strut injector is shown in Fig. 2 (right side). It has a leading edge angle of 45 degrees and an angle of 90 degrees at the trailing edge. This simplification of the injector allows two dimensional simulations for the case without fuel injection. In this case, the induced shock system can be effectively computed with high spatial resolution. With injection, only one thin three dimensional segment instead of the full strut geometry is used as the computational domain (Fig. 3).
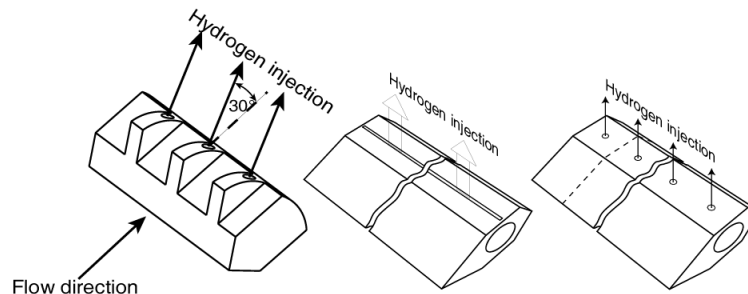

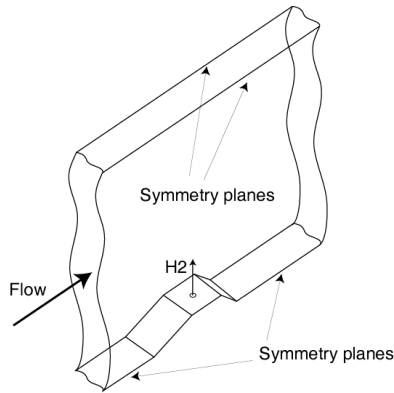
Figure 2: Injectors: pylon and struts

Simulation of flow in the chamber

Two dimensional simulation of the flow in the chamber were made for the "hot" case with the preheater in operation (P_0=7.5 *bar*, T_0=1300 *K*, M=2.15) in order to analyze the shock system and to obtain the zones with favorable conditions for the self-ignition. The contours of the Mach number and of the static temperature are shown in Fig. 4. The shock system produced by the strut increases the static temperature in the chamber. In the past, it was claimed in the literature that the flame is stabilized due to the temperature increase in the shock front leading to the ignition of the hydrogen. However, the maximal static temperature is achieved in the center of the chamber downstream of strut in the turbulent wake and at the intersection of the oblique shocks reflected at the chamber walls. The loss of kinetic energy downstream of the backward facing step dominates the process but the interference with the shock system is of additional importance. Thus, only in the wake the static temperature exceeds the ignition limit for the hydrogen and self-ignition cannot occur elsewhere. The expansion of the chamber after the strut leads to a drop of the static temperature below the ignition limit. The same temperature distribution was also obtained experimentally using the Raman spectroscopy for validation purposes [3]. It can be concluded that the wake downstream of the strut is the most important zone for the analysis of ignition processes and requires a detailed investigation.

Figure 3: Thin 3D segment used for the simulation

Investigation of the ignition zone

The ignition in the supersonic flow can be understood only through detailed studies of the mixing near the strut and in the turbulent wake downstream of the strut. The simulation of the flow in the chamber shows that a temperature of more than 900 *K* required for ignition is reached only in the wake behind the strut. In a first set of calculations of the reacting case, a preheated mixture of hydrogen and air was fed to the Laval nozzle in order to investigate the ignition process decoupled from the influences of fuel air mixing and the temperature field created by the low temperature of the injected hydrogen. This approach allows to make relatively fast two dimensional computations with high local spatial resolution although the detailed kinetic model [1] is applied.
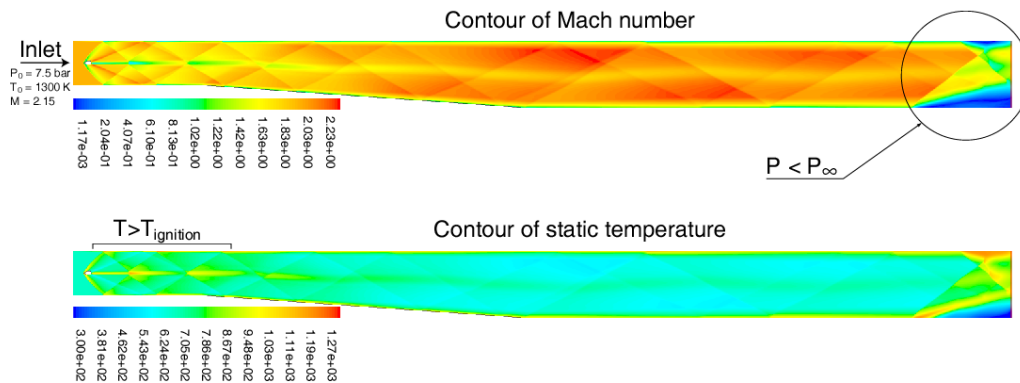

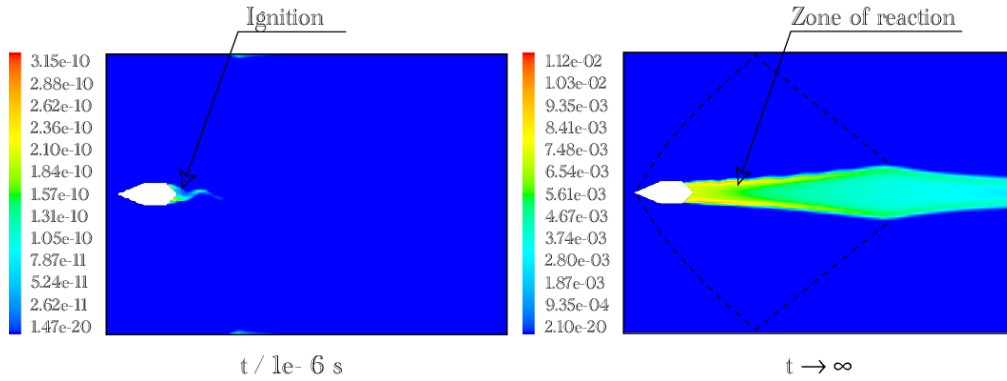Figure 4: 2D Simulation of the flow in the chamber

Figure 5: Simulation of the ignition and the development of the reaction zone (hydrogen mole-fraction)

As expected, the example with a stoichiometric ratio of $\phi = 1$ and a total temperature of 1300 K shows that the ignition occurs in the hot wake of the strut and the flame can not spread in transverse direction due to the lack of lateral transport through secondary vortices. The flame even converges after the interaction with the reflected shock from the wall due to the higher static pressure after the shock. Starting with the non reacted case as the initial condition for the time dependent calculation, Figure 5 shows the auto-ignition behind the strut and the development of the reaction zone from the ignition source. In the investigated case, the ignition results from the strong irreversible total pressure loss in the wake of the strut, while the shock wave system does not influence the ignition and the flame propagation. This result is in contrast with the statement found in the literature that reaction is induced and dominated by the shock system. This behavior is not specific to the geometry investigated, since a similar process will be found for other strut injectors that do not produce secondary flows.

Investigation of the influence of the inlet boundary conditions

In the former simulation the inlet boundary conditions were considered as ideal with constant flow parameters over the cross-section. In the experiment the simulation of the flight conditions is realized through the hydrogen preheater and the Laval nozzle which produce inlet conditions different from the ideal case because of the influence of the boundary layer. Thus, the real inlet profile differs from the ideal, may influence the shock system in the chamber and has to be taken into account for these reasons. While the averaged flow parameters such as Mach number, total and static temperature and pressure can be obtained from the experiment, the temperature and velocity profiles including the boundary layer are difficult to measure. Therefore, the inlet profiles had to be obtained numerically by simulation of the flow in the Laval nozzle. The geometry of the Laval nozzle is shown in Fig. 6. The simulation of the flow was realized for the cold and the hot mode of the preheater. In the cold mode the preheater is off and total temperature is 300 K, in the hot mode a total temperature of 1300 K was assumed. In both cases the total pressure is 7.5 bar. The profile of the Mach number downstream of the Laval nozzle is shown in Fig. 6. The asymmetry of the profile results from the asymmetry of the profile results from the asymmetry of the nozzle. The boundary layer reduces the Mach number from 2.15 in ideal case to 2.08.

Pseudo three dimensional simulation of flow near the strut

The mixing characteristic of the injector was analyzed using a three dimensional simulation in order to quantify the differences between slot injection and the injection by means of a line of orifices with the same flow area. The profiles obtained during simulation of the
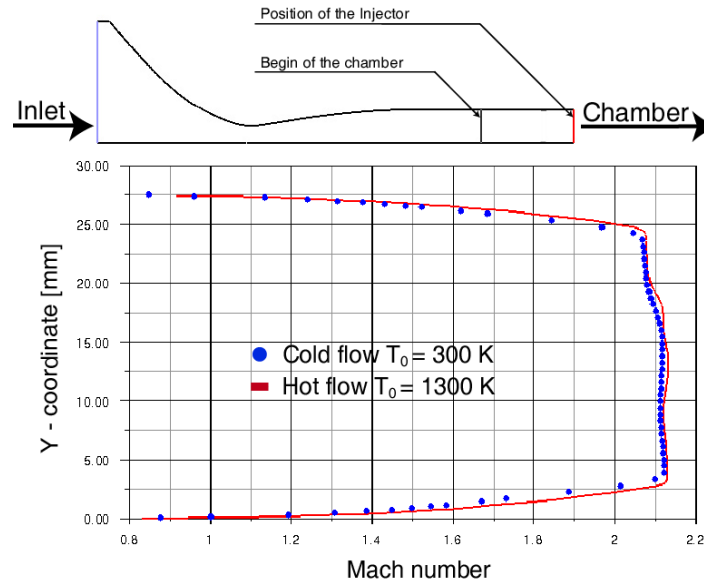
Figure 6: Geometry of the Laval nozzle and profile of the Mach number at the outlet of the nozzle

Laval nozzle were used to initialize the inlet boundary conditions. Because of the high degree of symmetry of the investigated geometry, a thin segment (see Fig. 3) can be selected as the computational domain as long as side wall effects are neglected. The result form this computation is shown in Fig. 7. The contour of static temperature is presented in color. It is seen that the oblique shock induced by the strut and the separation of the boundary layer resulted from shock boundary interactions. Because of the lower Mach number, the oblique shock is stronger compared to the case with constant inlet boundary conditions (Fig. 5). The boundary layers and their influence on the interaction with the shocks near the wall are clearly visible. The reflected shocks are shifted upstream and their interaction occurs closer to the trailing edge. Consequently, the static temperature and turbulence production are increased. This leads to the enhancement of the self ignition. Another effect of the shift upstream is that the interaction point lies in the subsonic wake of the strut. The higher static pressure produced by the shocks expands the wake and provides an ideal ignition region because of the high static temperature and low Mach number. In addition, the surfaces of the hydrogen jet are shown in Fig. 7. The fuel is located within the volume illustrated by the shaded surfaces. As expected, the lateral mixing is very weak because no secondary flow is induced by the strut and the penetration of the thin hydrogen jets is marginal.

The 3D simulation shows clearly that the potential ignition zones for the injected hydrogen are located in the wake of strut, where the local temperature and equivalence ratios are both favorable for self-ignition.

The 3D simulation with combustion showed that the ignition zone is located at the same place as in the test simulation of the premixed flow. However, a stable flame was not obtained, because no heat release is achieved despite an initial radical production in the vicinity of the strut (not shown). The analysis of this deviation form the behavior found in the experiment revealed several possible reasons. The simplification of the problem using a thin segment instead of real 3D geometry neglects the influence of the oblique shocks induced by the separation of the boundary layer at the strut's intersection with the side walls. These shocks influence the wake and enlarge it by increasing the pressure. Thus, the zone with high temperature is increased and this may lead to the flame stabilization in reality. Another reason
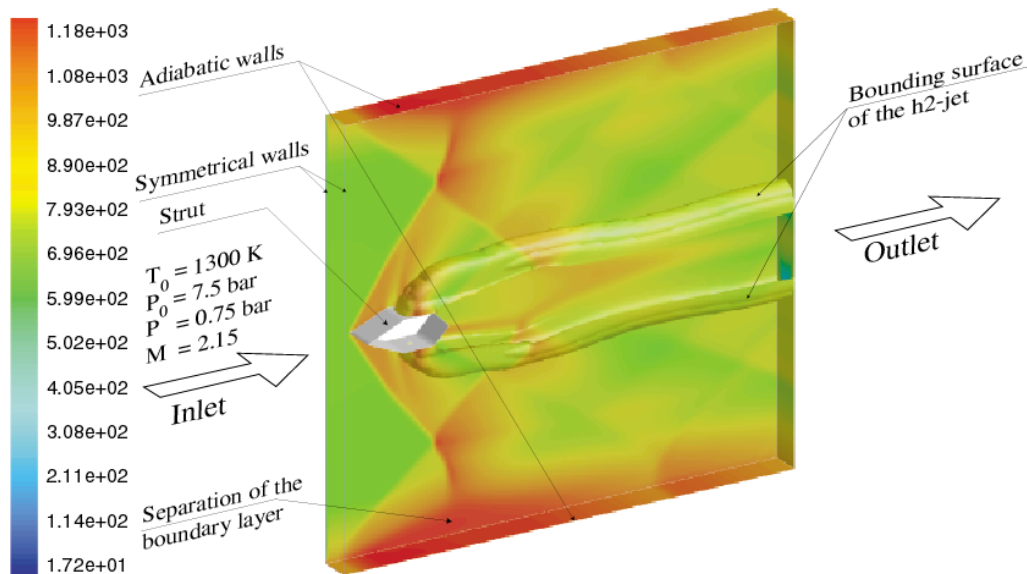
Figure 7: Three dimensional simulation of fuel injection near the strut: static temperature field and hydrogen distribution

may be the production of radicals in the hydrogen preheater used in the experiment which were not considered in the calculations so far. It is a well known fact that the influence of radicals on the ignition and flame stabilization can be tremendous even if their concentrations are low. These effects have to be investigated in the future.

Conclusions

The investigation of the shock system produced by a strut injector with subsequent mixing, ignition and flame stabilization showed that the characteristics of the ignition process is considerably different from the shock induced ignition which has been postulated in the past.

- The ignition is achieved through the irreversible total pressure loss in the wake of strut. The shock system does not significantly influence the ignition.
- The presence of a boundary layer in the chamber changes the shock system and the ignition conditions in the chamber considerably. The total pressure loss is tremendous increased due to the existence of boundary layers.
- The lateral mixing by the strut injector is very weak. This deteriorates both the ignition capability and the burnout.

REFERENCES

1. **Maas, U., Warnatz, J.** Detailed Numerical Simulation of H2-O2-Ignition in Two-Dimensional Geometries, Internal Report, Preprint 90-01, Universität Heidelberg, 1990.
2. **Gerlinger, P., Möbus, H., Brüggemann, D.** An Implicit Multigrid Method for Turbulent Combustion, Journal of Computational Physics, 2001, 166, 1-300
3. **Sander, T., Sattelmayer T.** Application of spontaneous Raman Scattering to the flowfield in a SCramjet Combustor, Intern. Conf. on the Methods of Aerophys. Research: Proc. Pt II. Novosibirsk, 2002.