Acta Astronautica 128 (2016) 540–550

Contents lists available at ScienceDirect

Acta Astronautica

journal homepage: www.elsevier.com/locate/aa

Effect of variation of length-to-depth ratio and Mach number on the performance of a typical double cavity scramjet combustor

Navin Kumar Mahto [a], Gautam Choubey [b,*], Lakka Suneetha [c], K.M. Pandey [b]

[a] Department of Mechanical Engineering, R.V. College of Engineering, Bengaluru, Karnataka, India
[b] Department of Mechanical Engineering, National Institute of Technology Silchar, Assam, India
[c] Department of Mechanical Engineering, Ellenki College of Engineering and Technology, Hyderabad, Telangana, India

ARTICLE INFO

Article history:
Received 25 July 2016
Received in revised form
12 August 2016
Accepted 13 August 2016
Available online 20 August 2016

Keywords:
Scramjet
Double cavity combustor
k–ε model
Length-to-depth ratio

ABSTRACT

The two equation standard k–ε turbulence model and the two-dimensional compressible Reynolds-Averaged Navier–Stokes (RANS) equations have been used to computationally simulate the double cavity scramjet combustor. Here all the simulations are performed by using ANSYS 14-FLUENT code. At the same time, the validation of the present numerical simulation for double cavity has been performed by comparing its result with the available experimental data which is in accordance with the literature. The results are in good agreement with the schlieren image and the pressure distribution curve obtained experimentally. However, the pressure distribution curve obtained numerically is under-predicted in 5 locations by numerical calculation. Further, investigations on the variations of the effects of the length-to-depth ratio of cavity and Mach number on the combustion characteristics has been carried out. The present results show that there is an optimal length-to-depth ratio for the cavity for which the performance of combustor significantly improves and also efficient combustion takes place within the combustor region. Also, the shifting of the location of incident oblique shock took place in the downstream of the H_2 inlet when the Mach number value increases. But after achieving a critical Mach number range of 2–2.5, the further increase in Mach number results in lower combustion efficiency which may deteriorate the performance of combustor.

1. Introduction

Scramjet engines are the future of hypersonic flights. With NASA's Hyper-X program [1] and successful flights of X-43A [2], air-breathing hypersonic propulsion appears very promising. But there are many challenges associated with the combustion in scramjet engines. Because of the very short time available for fuel injection, mixing, and combustion, a lot of research has been done to study the various flame holding mechanisms. The presence of normal fuel injector inside the combustor generates a detached normal shock towards the upstream direction of the injector. As a result, both the upstream and downstream of the injector, there is a formation of separation region which may influence the efficiency of the combustor. However, the downstream separation zones might be considered as a flame holder [3,4]. Though Normal injection technique provides recirculation regions and proper mixing of fuel and air inside the combustor, yet this technique also result in losses in stagnation pressure and an increase in drag characteristic of the engine [5].

Another alternative method for better-mixing phenomena in scramjet combustor is to use cavity flame holders. Studies have been shown that it can significantly improve the combustion as well as mixing efficiency in supersonic flows. The experimental observation regarding the comparison of different acoustically open cavities configurations to the baseline case with no cavity is performed by Yu et al. [6]. The results indicated that the profile of total pressure was more consistent at the outlet area and there was an enhancement in the volumetric heat release as well as in mixing phenomena near the cavity flow field of the combustor. Zhang et al. [7] studied the effect of the change in the length-to-depth ratio on drag and pressure fluctuations of the cavity. In his work, he found that the transverse mechanism controlled the oscillation in short cavity whereas; longitudinal mechanism controlled the oscillation in the long cavity inside the combustor. The computational work on the cavity flame holder with optimal sweepback configuration in scramjet combustor was mainly done by Huang et al. [8] and the computational result concluded that a cavity flame holder with 45-degree sweepback improved the performance of the combustion.

* Corresponding author.
E-mail address: gautam_dadaa@yahoo.com (G. Choubey).

http://dx.doi.org/10.1016/j.actaastro.2016.08.010
0094-5765/

Under several conditions, using of multiple cavities inside the combustor helps in producing better results than a single cavity. The dual cavity was shown to produce higher stream thrust and pressure ratio when compared to single cavity flame holder in the experiments performed by Collatz et al. [9]. Combustion characteristics in dual mode scramjet combustors have been studied experimentally by Micka et al. [10,11]. It was found from the experiment that for low values of temperature, the combustion would be concentrated near the leading edge of the cavity while it stabilized downstream in the jet wake for higher temperature values. An effective study on tandem dual cavity was done by Lu et al. [12]. In his work, he reveals that the presence of tandem cavity structure results in better flame stabilization and also helps in improving the combustion efficiency which is followed by shorter combustor length.

The numerical simulation of dual cavity scramjet combustor using Large Eddy Simulation can be seen in several works [13–16]. The use of two equation k–ε model [17,18] and SST k-ω model [19] to study combustion phenomena in scramjet combustor with strut injectors have also shown the effectiveness of these turbulence models in the cases of supersonic flows accompanied by combustion. Parameters like combustor aspect ratio, back pressure, Mach number etc. affect the combustion efficiency. The effects of the length-to-depth ratio and back pressure on cavity based scramjets have been studied numerically by Huang et al. [20]. To analyze these effects, the numerical model has to be solved a number of times with different configurations. The numerical investigation on fuel transport and mixing process in a scramjet combustor with rear wall-expansion cavity was performed by Zun et al. [21]. He also revealed that fuelling was an important factor in order to enhance the fuel mass fraction inside the cavity region. This process was mainly completed by employing a cavity direct injection on the rear wall of the combustor.

Ouyang et al. [22] worked on a parametric study of oscillation phenomena in combustion in a single side expansion scramjet and observed that distributed injection scheme is a productive method in order to avoid the combustion oscillation in scramjet combustor. Computational investigation on cavity structure of solid fuel combustor was reported by Xinyan Pei and Lingyun Hou [23]. They concluded that the decrease in total pressure loss inside the combustor was accompanied by the corresponding increase in cavity length, which was contradictory to the conclusion figured out in the case of liquid fuel scramjet. Wang et al. [24] performed LES studies of the interaction between jet and cavity in supersonic flow and noticed that mass exchange phenomena between the fluids in and out of the cavity were exceptionally increased by the jet–cavity interactions. As a result, there was a reduction in the residence time of cavity fluids. Though LES captures the flow better at high Reynolds numbers and computes the large scale mixing process very accurately [25], it consumes a lot of time and computational resources making it infeasible for analysis of several variations of combustor configurations. Instead, a simpler k-ε model can predict the model fairly accurately to study the effects of different changes in the inlet and geometric conditions.

Hence from the above discussion it can be noticed that there have been enough work regarding the physical phenomenon in the vicinity of the cavity, but the effect of variation of geometric configuration of cavity as well as incoming Mach number on the flow field of double cavity scramjet combustor is hardly discussed in specific.

The objective of this present work is to visualize the flow and combustion phenomena in a dual cavity parallel combustor using a k–ε turbulence model and also to study the effects of variation in the length-to-depth ratio as well as inlet Mach number on combustion performance of the present geometric configuration. At the same time, the validation of the present numerical simulation for double cavity has been performed by comparing its result with the available experimental data which is in accordance with the literature.

2. Flow modeling and simulation

2.1. Geometry and grid generation

The geometry of the 2D scramjet combustor used in the present analysis as shown in Fig. 1 is the same as used in the experimental study by Wang et al. [25]. It has a width of 50 mm and a height of 40 mm. The cavities are present both on the top and bottom walls of the combustor. Injection of hydrogen is done from a small cavity of 2 mm diameter located 10 mm before the combustor cavity. The top half of the geometry is modeled and symmetry condition is used along the centerline.

Because of the symmetry of the scramjet combustor configuration, the simulation needed only one half of the region to be performed for numerical analysis. In order to reduce the computational cost and design the combustor configuration efficiently, the 2 D computational configuration is used for the present geometry where all the meshing modeling and analysis are done using ICEM- CFD (Fig. 2) and Ansys 14- Fluent code [43] respectively.

Computational calculations of the flow were initially originated with a base grid of 110291 elements individually. The numerical grids were then clustered in the region of boundaries. These grids were then adapted based on gradients of static pressure for the enhancement of mesh and also to catch the shocks which were further resulted in improvement close to, boundary layer on the wall [19]. For solving the mixing process and diffusion impact appropriately, the refinement of the mesh resumed between the incoming air and hydrogen jet in the shear layer portion. On the other hand, in order to determine the combustion phenomena in the stream field precisely, meshes were further refined based on the gradients of reacting species. Finally, unstructured meshes with triangular 717,900 elements are adopted as final mesh for 2D geometry for all subsequent calculations as shown in Fig. 2.

2.2. Grid independency analysis

For the case of grid convergence study in the present work, variation of static pressure along the top wall is described along with three different mesh scales namely the coarse mesh(110291 nodes), the moderate mesh (354,321 nodes) and the refined mesh (717,900 nodes). The maximum pressure rise for 717,900 and 929,789 elements are nearly alike without any noteworthy changes which is clearly noticed from Fig. 3. The computational results are under-estimated in some regions but overall satisfactory result is obtained with the experimental data. It is clear that the grid scale makes only a marginal difference to the wall pressure distribution. On the other hand, in order to obtain a steady flow field, there is an increase in the number of time step along with the increase in the number of grid cells and this is applicable to the parallel computing environment and the computational method used as well[26]. At the same time, proportionality is observed between the accumulation of stochastic error and the number of time steps (The accumulation of stochastic error $\propto$ to the number of time steps). The accumulation of stochastic error also relies on accuracy of the scheme and approximation error [27] and this is out of the scope during this research work. Thus, this discussion has not been enclosed within the current study.