SIMULATION OF SCRAMJET COMBUSTOR USING QUASI 1-D AND 3-D CFD MODELS

Ch Naga Nitish[a], Malsur Dharavath[b*] and P Manna[b]
[a]M Tech student, Department of Mechanical Engineering, Amrita Vishwavidyapeetham, Amritapuri [b*/b]Scientist, Directorate of Computational Dynamics, DRDL, Hyderabad

ABSTRACT

The success of a hypersonic vehicle will depend upon the development of an efficient propulsion system that is able to produce sufficient thrust. Hydrogen fuel stands in first place among other fuels, when fuel specific impulse is considered[1]
.

*In the preliminary design of scramjet propulsion system for a hypersonic vehicle, it is of a great significance to accurately predict the engine performance with high computational efficiency. The concept of mixing controlled supersonic combustion is presented by Heiser and Pratt [**2**].*

Birzer and Doolan had developed a quasi-one-dimensional solver that incorporates mixing model only for strut injectors and tried to apply the same model for a scramjet combustor with wall injectors by assuming mixing length that fits the experimental data. However, it is difficult to always assume the right mixing length. Therefore, it is ideal to consider a mixing efficiency model that is suitable for wall injectors. To comprehensively predict the scramjet engine performance in quick time and provide basic support for the development of the engine, a quasi-onedimensional solver with the consideration of the mixing model for wall injection is developed in the present work[3]

*The schematic representation of the scramjet combustor (Yu et.al.), considered for the present study is shown in Fig. 1. The combustor is having a nearly constant are section of 295 mm length followed by a divergent section (both sides) of 1.5^o of 300 mm length. Gaseous hydrogen fuel in injected from top and bottom walls at 140 mm distance from combustor entry. To check the credibility of the solver, 3D simulations are carried out to calculate thrust and compare it with the thrust predicted by the quasi-one-dimensional model. The geometry has 1/4th symmetry and therefore only quarter geometry is considered in present case to reduce the computational time. The computational domain of scramjet combustor is shown in Fig.2. Multi-block structured grid is generated using ICEM-CFD software. The X, Y, Z axis are taken along the length, height and width of scramjet combustor respectively. Origin is place at the middle point of the combustor entry. 3D numerical simulation is carried out using commercial ANSYS CFX-17.2 software [**5**] at facility nozzle entry condition of P_s = 7.1 bar, T_s = 1642 K, u = 813 m/s and hydrogen fuel equivalence ratio of 0.46.[4]*

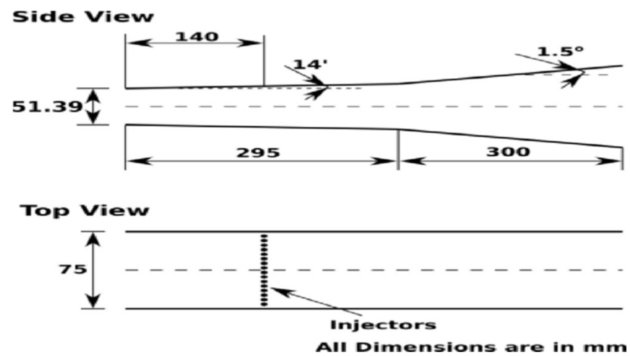

FIGURE 1. SCHEMATIC REPRESENTATION OF SCRAMJET COMBUSTOR [**4**].

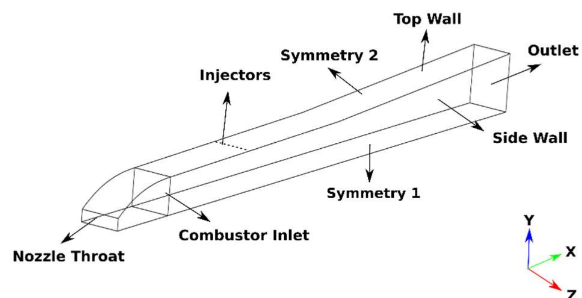

FIGURE 2: COMPUTATIONAL DOMAIN OF SCRAMJET COMBUSTOR.

QUASI 1-D MODEL

The quasi-one-dimensional model is constructed by using the governing equations taken from Doolan and Boyce [**6**] and the mixing model is taken from Heiser and Pratt [**2**]. Mach number, pressure and temperature distribution along the combustor length obtained by using the current solver is shown in Fig. 3. From the figure, it can be said that the pressure and temperature start to rise from the point of injection and then decreases in the diverging section of the scramjet combustor. From the Mach number distribution, it can be said that the flow is fully supersonic throughout combustor.

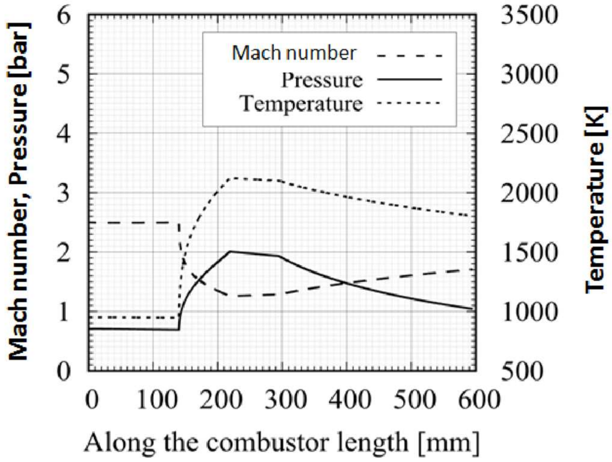

FIGURE 3: MACH NUMBER, PRESSURE AND TEMPERATURE DISTRIBUTION ALONG THE COMBUSTOR LENGTH.

3D SIMULATION

The 3D simulation is carried out with the help of ANSYS CFX-17.2 [5], which is a cell based finite volume solver. Grid independence study has resulted an optimum mesh size of 16 million nodes. The Mach contours at the mid-width of the scramjet combustor is shown in Fig.4. From the Mach contours, it can be said that a recirculation zone is formed near the injection location and subsonic pockets are visible in this region.

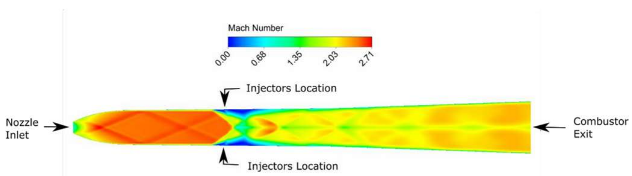

FIGURE 4: MACH NUMBER CONTOUR AT MID-WIDTH OF SCRAMJET COMBUSTOR.

COMPARISON OF QUASI 1-D AND 3-D CFD MODEL RESULTS

The results of the pressure distribution obtained from the current quasi-one-dimensional solver, 3D simulation and the experimental data from Yu et.al. [**4**] are compared in **Fig.5**.The pressure before injection region, obtained from experiment and CFD simulation, is high when compared with the pressure distribution obtained from quasi-one-dimensional model. This is due to the boundary layer separation upstream of the injectors. Since the current model does not account for boundary layer separation, pressure is poorly predicted in that region. However, the model has predicted the pressure with high accuracy after attaining the maximum pressure, which is crucial for calculating thrust. The thrust distribution obtained from CFD simulation and quasi-one-dimensional model is shown in **Fig.6.** The thrust obtained from both the calculations matches fairly well. Hence, the model is able to calculate the thrust with reasonable accuracy with much lesser computational effort. In conclusion, it can be said that the current quasi-one-dimensional solver is useful in predicting thrust in the preliminary design of scramjet combustor to obtain faster meaningful results.

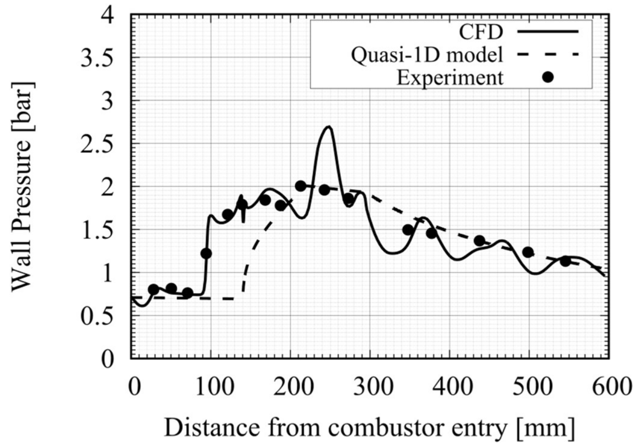

FIGURE 5. COMPARISON OF TOP WALL PRESSURE DISTRIBUTION ALONG THE COMBUSTOR LENGTH.

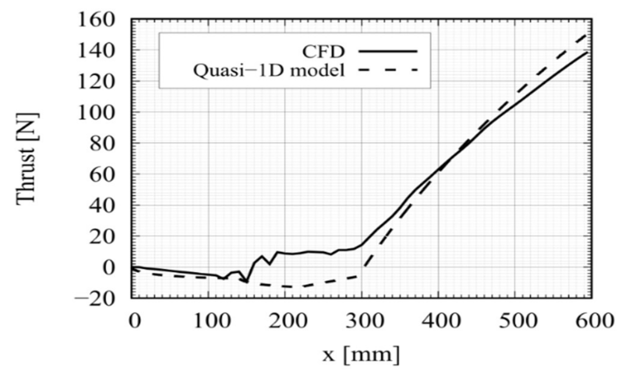

FIGURE 6: THRUST COMPARISON BETWEEN QUASI-ONE-DIMENSIONAL MODEL AND 3D CFD SIMULATION .

REFERENCES

[1] E.T. Curran, S.N.B. Murthy, Scramjet Propulsion. Progress in astronautics and aeronautics 189 (2000).

[2] W. Heiser, D. Pratt, Hypersonic Air-breathing Propulsion. AIAA: Education Series (1994).

[3] Cristian Birzer, Con J. Doolan, Quasi-One-Dimensional Model of Hydrogen-Fueled Scramjet Combustors, Journal of Propulsion and Power London 25: 1220-1225 (2009).

[4] G. Yu, J. G. Li, X. Y. Zhang, L. H. Chen, B. Han, C. J. Sung, Experimental Investigation on Flameholding Mechanism and Combustion Performance in Hydrogen-fueled SupersonicCombustors. Combust. Sci. and Tech. 174: 1−27 (2002).

[5] ANSYS CFX-17.2 commercial software, released in 2016.

[6] Doolan, C., and R. Boyce, R., “A quasi-one-dimensional mixing and combustion code for trajectory optimisation and design studies”. 15th AIAA International Space Planes and Hypersonic Systems and Technologies Conference, 04, pp. 1–10, (2008).