International Review of Aerospace Engineering (I.RE.AS.E), Vol. xx, N. x
(Accepted Paper) January 2020

Flow-Field and Performance Study of Coaxial Supersonic Nozzles Operating in Hypersonic Environment

Mohammad Samara [1], Ashish Vashishtha [2], Yasumasa Watanabe [3], Kojiro Suzuki [4]

Abstract – The integration of multiple propulsion systems in a coaxial configuration is one of the challenges to realize hypersonic passenger transportation. This study is an attempt to understand the performance of two co-axial jets exiting from the base of slender body, operating in single and dual operation mode in freestream hypersonic flow environment. In addition, the effects on performance by adding a different length common channel to both co-axial jets are studied. In the first part of this study, the experiments have been performed for small slender body kept in hypersonic Mach 7 flow, which consists of two inner high-pressure chambers and two co-axial nozzles at the base: central nozzle (Mach 4) and surrounding nozzle (Mach 2.8) along with extended common region, termed as common channel. Schlieren images have been captured for single and dual operation modes. Axisymmetric numerical simulations have been performed for further understanding of the flow interactions and have been qualitatively validated with experimental images. In the second part, the parametric study has been performed using numerical simulations for resized model with various exit Mach numbers for central and surrounding jets along with effect of no common channel and with common channel for various operation modes. One of the findings of the study is that dual jets should operate and exit at same plane (no common channel) with the same exit area (each nozzle with half of total available exit area) in order to have higher total thrust from both jets than the sum of individual jets operating in single operation mode. For higher central jet Mach numbers, the corresponding surrounding jet Mach number will be lower, and in dual operation mode (without common channel), the total thrust will be the same or lower than the sum of the individual jet operations. Regarding the effect of common channel, it has been found out that the introduction of the extended short or long common channel in dual mode operation does not have significant effect on thrust, while the jet flow field is strongly affected by the common channel presence. In single operation mode, for Mach 2 central-jet, the thrust performance decreases 12.2-14.6 % in presence of short and long (29.5 mm and 59 mm) common channel, while for Mach 2 surrounding jet, the thrust performance increases by 15-17.4 % in presence of common channel.
***Keywords**: Hypersonic Flow, Supersonic Jets, CFD, Wind Tunnel Experiment, Co-axial Nozzles*

Nomenclature

P	: Pressure
ρ	: Density
R_e	: Reynolds number
T	: Temperature
$\dot{m}$	: Mass Flow Rate
P_0	: Stagnation Pressure
M_{exit}	: Exit Mach Number
V_{exit}	: Exit Velocity
P_{exit}	: Exit Pressure
A_{exit}	: Exit Area
P_a	: Surrounding Pressure
u	: x-component of velocity
v	: y-component of velocity
x	: x-cartesian coordinate direction
y	: y-cartesian coordinate direction
D	: Diameter
n	: number of mesh nodes in x-direction
m	: number of mesh nodes in y-direction

I. Introduction

For the development of future hypersonic passenger transportation, it is required to develop air-breathing propulsion system that can operate in flight stages at various Mach numbers ranging from subsonic, supersonic through hypersonic speeds. It can be achieved by combining two or more propulsion systems. A simple configuration for combining multiple propulsion systems can be the co-axial engine configurations. Such configurations have been applied historically in the SR71 black bird engines [1], which integrate the Pratt and Whitney J58 engine with a ramjet engine one. Co-axial configuration has been also proposed for the hypersonic air breathing flight as the Synergistic Air-Breathing Rocket Engine (SABRE) [2]. In the proposed SABRE engine configuration, an air-breathing rocket engine is combined with ramjet as an axisymmetric co-axial configuration. However, there are very few fundamental studies available in literature of single or dual mode operations of combined co-axial nozzles in hypersonic environment and its impact on thrust performance.

Manuscript received and revised December 2019, accepted December 2019

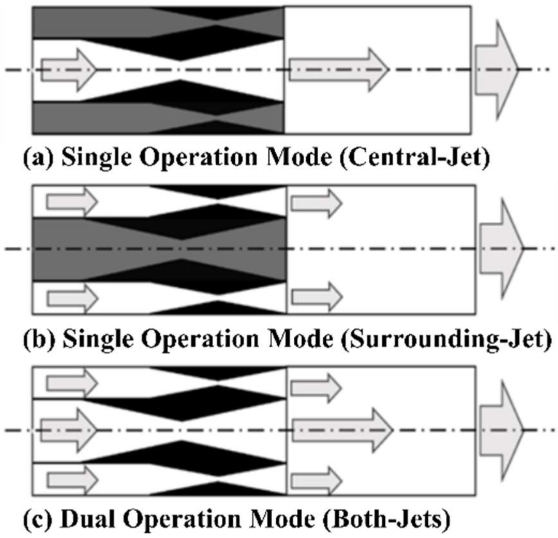


Figure 1. Coaxial configuration and three operational modes with common channel.

This study focuses on the experimental and numerical investigation of co-axial supersonic air jets discharging into hypersonic flow environment from the base of axisymmetric slender body in single (either central or surrounding jet) or dual (both jets) operation modes. In addition, in order to study the augmentation of the total thrust from both the nozzles, the walls of the outer nozzle have been extended so that central and surrounding jets interact in a confined passage before exiting into the low-pressure hypersonic external flow. In this study, this confined passage is termed as common channel.

Figure 1 depicts the single and the dual operation modes for co-axial nozzles with common channel. When a single supersonic jet exits at the base of hypersonic vehicle, low environmental pressure at high altitude would cause the supersonic jet to be under-expanded most of the time during its flight and subsequently leads to a loss of partial thrust due to the need of further expansion. The extended straight passage can reduce the expansion level, which may lead to the improvement in the thrust performance. However, extending the length of the straight passage can also add the weight and can increase the length of vehicles.

The longer extended common channel can also lead to reflection of waves on the wall, including expansion front, depending on the expansion level and Mach disk may also form before the end of common channel, which may lower the performance of the system. Hence, it is important to understand the effect of common channel length on thrust for single (either central or surrounding only) and dual operational modes to optimize the length. It is expected that the optimum length of common channel can improve thrust performance by reducing the expansion level in each single operational mode and provide some common region to augment thrust in dual operation mode.

Historically, studies on supersonic axisymmetric over-expanded, correctly expanded, or under-expanded single jets have been extensively conducted at free or open-jet facilities by discharging it into static atmospheric conditions [3]. One of the studies of co-flowing supersonic over expanded jets with central Mach number 1.96 have also been performed in open-jet facility [4] and it has been found out that the jet core length increases in presence of co-flow. In this study, with the increase in surrounding jet speed, under-expanded jet interaction with supersonic overexpanded central-jet, complicated interactions among the shock-cells has been observed and jet field has been modified significantly. Murli et al. [5] have studied the interaction of twin sonic jets in parallel and inward canting configuration. Safir et al. [6] have investigated laminar to turbulent transition of coaxial subsonic jets discharging into static air. In few studies, under-expanded jets in the hypersonic regime have been numerically investigated [7] by solving the Navier-Stokes equations applied to hypersonic exhaust plume / afterbody flow fields. Another interesting study that simulates a single under-expanded jet [8] in supersonic environment utilizes the PIV technique to visualize the flow field of the jet as well as its interaction with the free stream flow and the wake behind a body facing a Mach 3 flow condition. A recent study on the Skylon Airplane and its SABRE engines Plumes [2] has simulated the full body of the airplane along with the effect of its propulsive jets on the airplane aft-body at various Mach numbers 5 to 17. Stephan et al. [9] at DLR, have studied scaled down model of Ariane launcher exiting Mach 2.5 flow of air and Helium gas to represent plumes, in a hypersonic Ludweig tube operating at Mach 5.9. Clifton et al. [10] have conducted measurements for co-axial jets of Mach 1.8, with central jet as argon and co-flow as air in open atmosphere in order to validate the CFD code for mixing studies. In a different study, Baurle et al. [11] have studied mixing of Helium-air and argon-air for same co-axial nozzles of Clifton et al [10] exiting in open atmosphere by using hybrid RANS/LES simulations. Zaman et al. [12] have conducted experimental and computational study of various tones occurring in supersonic designed co-axial annular nozzles operating in range of low Mach numbers. Cliff et al. [13] have investigated the wind tunnel model design to study the effect of nozzle exhaust on sonic boom using CFD for different exhaust nozzle pressure ratios. Recently, Ivanov et al. [14] experimentally investigated hydrogen fueled detonation ramjet model in a pulsed wind tunnel with air streams of Mach numbers of 4, 5, 6, and 8. In this experiment, supersonic airflow enters in an annular combustor, and stable and oscillating modes of detonation have been obtained. From the above discussion, it can be drawn that there are very few studies conducted about aircraft and propulsive system integration in wind tunnel environment and further integration of two or more propulsive system in high-speed wind tunnel environment are rarely found.

Apart from the effect of surrounding flow-field on the exhaust jet, the performance of co-axial supersonic jet depends on common channel, which can be theoretically considered as sudden expansion region because of available downstream area increase after exiting from nozzles in single and dual operation modes. When the single jet is exiting from the base of slender body in very low-pressure environment (similar to pressure and temperature of hypersonic free stream), and there is no flow around the body, it will be highly underexpanded. However, when there is flow around the body, the exiting jet will interact with the shear layer formed because of the flow separation at the corner before interacting with low-pressure region after the expansion waves at the corner, this will lead to reduction in underexpansion level. Since this study is focused on the hypersonic flow environment, the momentum of hypersonic flow around the body will be higher than the supersonic one, the strong shear layer may play important role in reducing the underexpansion level. The common channel may provide region for mixing in dual operation mode, which may turn the flow in freestream direction and may result in better thrust performance. Supersonic flow exiting in suddenly expanded region has been studied by Bulat et al. [15] and classified into different flow regimes as steady, oscillating and transient, based on the interaction with wall and re-circulation regions. Further, the effects of suction and blowing on flow Separation in a symmetric suddenly expanded channel have been also examined by Layek et al. numerically [16]. Suddenly expanded nozzle is well studied for the case of single jet expanding in higher area channel. However, the effects of surrounding jet on a suddenly expanded central jet in hypersonic environment have not been studied in the literature according to the knowledge of authors. This study will evaluate the performance of circular central, annular surrounding jet exiting in the common channel area as well as both jets simultaneously exiting in common channel region before interacting with hypersonic flow environment.

The main objective of this study is to evaluate the performance of the co-axial nozzle in a hypersonic environment and to study the effect of the additional common channel, whether it enables thrust augmentation for various operation modes. The whole study is divided in three parts. In the first one, the experimental study is conducted for small slender body with co-axial jets exiting from the base by qualitative schlieren flow visualization of shock structure for single and dual operation modes with common channel. Single point wall pressure measurements are also performed in order to validate CFD quantitatively. The axisymmetric numerical simulations are performed for the same experimental cases to evaluate the performance of system by thrust calculations. The numerical simulations are validated qualitatively by flow structures as well as quantitatively by single point wall pressure measurements from experiments. In section III.3 qualitative and quantitative CFD validations are explained. In the results (section IV.1), the general flow field with various shock and expansion wave interactions have been discussed for the single and dual mode operations from experiments and CFD for experimental model. In the second part of study, section IV.2.1 describes the parametric study of enlarged experimental model (without common channel) by isentropic 1-D calculations as well as axisymmetric CFD simulations, operating at various central and surrounding jet Mach numbers without common channel. The effect of the operating exit Mach numbers on the performance in single and dual operation modes have been studied with the thrust evaluation and flow-field analysis. In the third and last part, the effect of the addition of various common channel lengths on the flow-field and the thrust performance of the system have been evaluated by numerical simulations. Section IV.2.2 compares the effect of common channel lengths on thrust performance in different operating modes. Sections IV.2.3-IV.2.5 describe the flow field and provides explanation of variation in thrust in two single (each nozzle) and dual operation modes.

II. Experimental Method

II.1. Hypersonic wind tunnel

Table1. Hypersonic wind tunnel specifications [17]

Technical Specification	Values
Mach Number	7.0
Stagnation Pressure	Maximum 0.950 MPa
Stagnation Temperature	Maximum 1000 K
Mass Flow Rate	Maximum 0.39 Kg/s
Nozzle Exit	200 mm diameter
Run Time	60 sec. (maximum)

Initially, experimental studies have been conducted to visualize the basic flow structure in various operation modes. The experiments have been conducted in Kashiwa Hypersonic Wind Tunnel Facility at Graduate School of Frontier Sciences, The University of Tokyo [17]. The wind tunnel has a uniform jet core of 120 mm diameter of Mach 7. The size of the jet-core limits the experimental model size and the use of intrusive measuring instruments that can be installed inside the test section. The general specifications of the hypersonic wind tunnel are shown in Table 1. In all the experimental cases, the stagnation pressure has been kept at 950 kPa, whereas the stagnation temperature has varied from 500 K to 600 K, but it is assumed to be approximately 600 K for numerical studies.

II.2. Experimental set-up

The overall design of the experimental model is shown in Fig 2. All the components of the model have been fabricated in-house at the wind tunnel machine shop, the University of Tokyo. The experimental model is designed to be compact and able to generate supersonic coaxial jets, with measuring capability of its settling chamber pressures. It should be noted here that the nozzle walls of the model are polished to be as smooth as possible.

The total length of the experimental model is 121 mm and the maximum cross-section diameter of slender body is 16 mm. The experimental model has been fixed by a threaded bolt assembled to the moving sting in the test-section, along with the inlet pipes for settling chambers and pressure measurement tubes. The multiple step front nose has been used to reduce the overall length of the experimental model so that whole model should be inside the area of optical window for visualization. The internal features of the experimental model (Fig. 2) consist of two

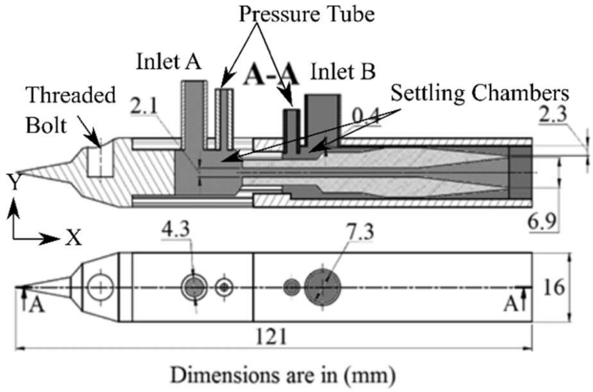


Figure 2. Experimental model components.

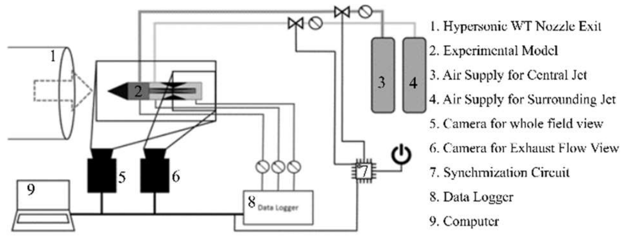


Figure 3. Experimental setup inside the test section.

high pressure settling chambers, which supply air to central jet and surrounding jet individually. The settling chambers are connected through the inlet pipes A and B, from which the air can be supplied during wind tunnel operation at constant pressure. Both the settling chambers are connected to pressure tubes as well as pressure sensor in order to ensure constant pressure is supplied during the operation of jets. In order to adjust the exit of both the convergent –divergent nozzles at same axial location (or adding common channel), the throat of the central jet is elongated.

The design specifications of both the central-jet nozzle and the surrounding jet nozzle are tabulated in Table 2. The throat diameter of central jet is 2.1 mm, while the throat annular channel outer diameter of the surrounding jet is 12.3 mm with an internal channel diameter of 11.5 mm. The internal exit diameter of the common channel is 12.3 mm. The design of the central jet nozzle is considered to have exit Mach number as 4, taken in reference to Martini et al. [18] which is one of the operating conditions for rocket engine in a hypersonic environment. The surrounding jet exit Mach number has been designed to represent scramjet engine exit Mach number 3.4 operating point according to Tatum et al. [19], but due to the limitation of the length of the experimental model and the available exit area, the surrounding jet has been designed with exit Mach number 2.68, which is also reasonable exit Mach number condition for scramjet operation. The wall thickness between central and surrounding jet nozzle at exit is 0.4 mm. The exits of the central and the surrounding nozzles have common channel area (of same cross-section) with length of 5.4 mm.

Table2. Central and surrounding nozzle design characteristics

Items	Central Jet	Surrounding Jet
Throat Area	3.46 mm^2	14.95 mm^2
Area Ratio	10.896	3.1132
Exit Mach Number	4.0	2.68
Isentropic Pressure Ratio	0.0066	0.044
Isentropic Temperature Ratio	0.2381	0.410
Common Channel length	5.4 mm	
Common Channel Diameter	Internal: 12.3 mm	

Figure 3 shows the overall experimental set-up, which consists of high-pressure reservoirs to supply air, air pressure regulator, cut-off valves, flow connecting tubes, and pressure measurement tubes. The experimental set-up has been equipped with schlieren system in order to visualize the external flow field. The system uses two cameras: the first one captures the flow around full body and the second one captures the jet exhaust area. Synchronization mechanism is used to capture schlieren images from two cameras as well as pressure measurements from settling chambers of experimental model, simultaneously. During the experiments, the high-pressure reservoir is connected to experimental model settling chamber by a pressure tube through pressure regulator and cut-off valve for both central and surrounding jets, separately. The sequence of the events during experimental run is as follows:

1) Hypersonic tunnel starts first and when the flow is established in the test-section, the experimental model is inserted to the flow at zero angle of attack.
2) Pressure valves are opened to supply air into settling chambers of either central or surrounding jet or both.
3) Schlieren set-up and pressure measurements are triggered to capture images and collect data.
4) Flow stops in hypersonic tunnel and air supply stops from the high-pressure reservoir to the settling chambers.

The overall run time for the wind tunnel operation has been approximately 30 s. During this run time, all the above-mentioned operations are conducted and pressure and schlieren data have been captured.

Many experiments have been conducted by varying settling chamber pressure for central and surrounding nozzle with single common channel length (5.4 mm). However, for this study only three data point (one for duel operational mode, and two for single operational mode; central jet only and surrounding jet only) have been used to validate CFD results qualitatively by comparing flow features captured using Schlieren system. Several other settling chamber pressures of central and surrounding jet combination are used to compare measured and computed pressure at 2.8 mm internal location from the exit of common channel. The details of these operating conditions are mentioned in the section III.3.

III. Numerical Method

The experiments in hypersonic wind tunnel provide limited data to analyze the performance of the system because of small experimental model in the test-section and involving intrusive measurements may lead to affect the flow field. Hence, the qualitative measurement data from schlieren images are used to validate axisymmetric numerical simulation. The performance of the co-axial jets in hypersonic wind tunnel along with slender body has been accessed from numerical analysis. In this section, the numerical procedure is explained along with qualitative comparison with experiments, where the computational conditions are set to reproduce the experimental conditions.

III.1. Computational Domain

The computational domain has been composed of two dimensional structured, multizone grid. The overall domain has been divided in six different zones as shown

Zones mesh size:
Zone A: n: 52, m: 101 Zone C: n: 33, m: 101 Zone E: n: 53, m: 401
Zone B: n: 51, m: 801 Zone D: n: 51, m: 801 Zone F: n: 101, m:1001
Total number of grid nodes = 224191

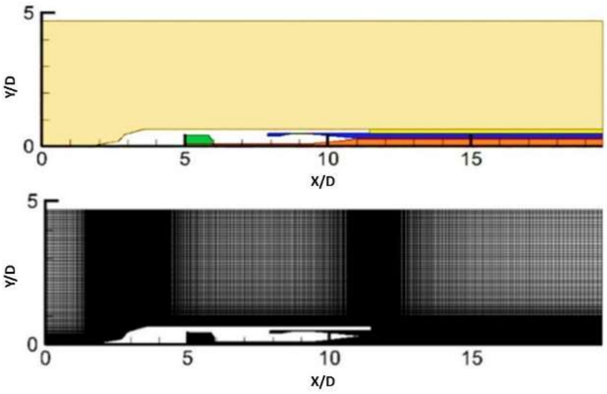


Figure 4. Multi-zone computational domains

in Fig. 4. A two-dimensional grid has been generated for each zone by keeping the same distribution of points at each interface. The minimum mesh size has been of the order of 1×10^{-6} m, with the mesh refinement imposed near the walls. The overall computational domain has 224191 nodes as shown in Fig. 4. The grid sizes are set to be small enough to capture the shock wave structures, and the boundary layer velocity gradient near the walls of the nozzle as well as on the body wall. The final grid size is obtained by performing grid independence study for three coarse, medium, and fine size and medium one as shown in Fig. 4 had been chosen for this study by comparing the shock location and thrust.

III.2. Numerical Schemes

The numerical simulations are performed by solving the generalized axisymmetric Navier-Stokes equations for unsteady compressible laminar flows. The inviscid flux in each direction is computed using Liou's all speed AUSM+ up scheme [20] with upwind biased third order MUSCL interpolation as well as entropy fixation. The viscous flux in each direction is computed by 2nd order central difference scheme. The third order 3-step TVD Runge-Kutta method [21] has been used for time marching. The inlet boundary conditions are used in the numerical simulations at inlet as wind tunnel test-section condition and at outlet extrapolated from inner domain. On the wall adiabatic no slip boundary condition is used while freestream and axisymmetric boundaries are imposed at appropriate locations.

The numerical simulations have been performed for different experimental cases by changing the settling chamber pressure of central jet and surrounding jet for same common channel length. In the later part of study, the parametric study has been performed for enlarged model. The central jet Mach numbers have been varied for parametric studies and accordingly surrounding jet Mach number changes, while the total exit area remains the same for resized models. The common channel length has also been varied for parametric study in order to understand the modification of flow field as well as thrust performance in presence of common channel.

III.3. CFD Code Validation

In this section, initially, the CFD results obtained for single and dual operation mode for experimental model have been compared qualitatively with experimental flow visualization. In the second set of experiments, the wall pressure at 2.8 mm inside from the common channel exit has been measured for various combination of settling chamber stagnation pressures and compared with CFD results for validation.

Qualitative Comparison:

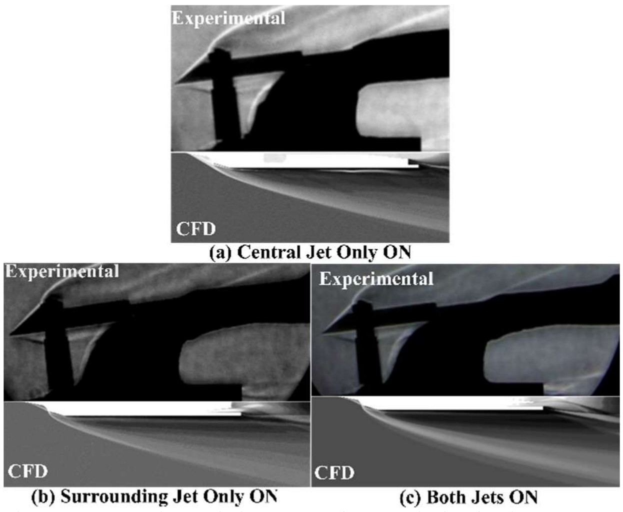


Figure 5. Experimental images and numerical (density contour) comparison (a) central jet (401 kPa), (b) surrounding jet (145 kPa) and (c) both jets active (395 kPa / 195 kPa)

The CFD results for single and dual operation modes have been compared with related experiments qualitatively in term of shock formation and exhaust jet flow features. Figure 5 compare the shock formation from experiments and CFD for outer body (a) only central jet (settling chamber operating pressure 401 kPa) (b) only surrounding jet (settling chamber pressure 145 kPa) and (c) both jets operating (settling chamber pressure: central jet 395 kPa, surrounding jet 195 kPa) in dual operation modes. In Figs. 5, the experimental model has been mounted on a structure and has different shock wave formation on the upper part because of complicated experimental set-up than CFD. However, there is no significant difference in the qualitative jet structure, when compared individually with experiments. Flow features such as common channel internal oblique shock waves, trailing shock wave and jet-boundaries are qualitatively reproduced successfully in CFD simulation. The jet-flow field will be discussed in detail in the result section.

CFD model assumes flow is laminar, which seems to be reasonable for current study as Reynolds number of cold flow (T_0=300 K) exiting from the nozzle of common channel is of the order of 10^4 (7.25×10^4) and nozzle wall is fabricated to have a smooth surface. It can be said that CFD can adequately reproduce the jet-flow field in hypersonic environment and absence of the exact upper geometry of experimental set-up would not affect the purpose of the study. The thrust is calculated from CFD simulations. It is found out that the calculated thrust during single operation modes is 1.793 N and 2.187 N for central and surrounding jets, respectively. However, during the dual operation mode, when both jets are active, the calculated thrust is 4.691 N.

Quantitative Comparison:

In order to validate CFD simulations quantitatively, single point pressure measurements have been conducted. The pressure tap (1 mm diameter) on the wall of common channel has been located at 2.8 mm inside the exit. A set of MPX2010 Silicon Piezo-resistive pressure sensor has been used with +/- 0.1 kPa accuracy. Various single and dual mode operations have been conducted as listed in Table 3. The central jet design Mach number is 4.0 and the surrounding jet design Mach number is 2.68. These experiments lead to modification in the experimental set-up; hence they have been conducted separately in the end of experimental campaign. Further CFD simulations are performed for three dual mode and two single mode (either central jet or surrounding jet) operations with boundary conditions same as experiments. The measured and the computed wall pressures are compared in Table 3 along with error between them.

Table3. Comparison between Measured and Calculated Wall Pressure

P_0 (Central-Jet) (kPa)	P_0 (Surrounding-Jet) (kPa)	Measured Pressure (kPa)	Computed Pressure (kPa)	Error (%)
545.31	69.373	1.831	1.568	14.36
543.47	67.662	1.778	1.526	14.17
364.90	61.907	1.642	1.338	18.51
577.53	None	0.478	0.387	19.04
None	107.48	2.022	2.128	5.24

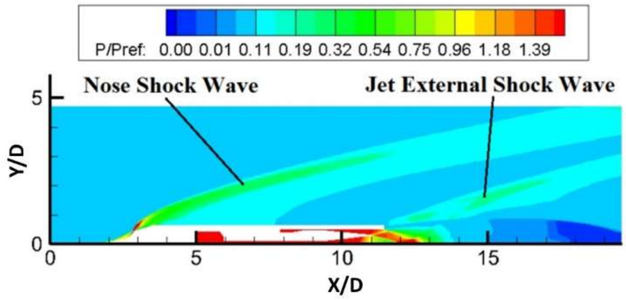
The difference in measured and computed pressure is less than 20 %, which can be attributed to some differences in shape between experimental model and computational model as the throat diameters in experimental model is very small in size. However, the trend of the computed pressure and the measured pressure shows consistency and validate that CFD simulations can be utilized for studying the parametric studies on performance of co-axial nozzle in hypersonic environment.

It can be concluded here that current CFD set-up can reproduce the general flow features of co-axial nozzles in hypersonic environment as well as predict the trend of pressure measurements in comparison to experiments to utilize them to study the performance of such systems.

IV. Results and Discussions

In this section, the flow features of co-axial supersonic jets in hypersonic environment are discussed regarding single and dual mode based on experiments and CFD simulations. In the second part, the parametric study has been performed for enlarged model by isentropic calculations as well as CFD to study the effect of different nozzle exit Mach number without common channel. In the final and third part, the effect of common channel length on the performance of co-axial supersonic system in hypersonic environment has been analyzed using CFD simulations.

(a) Experimental schlieren full body for dual operation mode



(b) CFD pressure contours for axisymmetric simulation

Figure 6. Flow around the body with coaxial jet (a) Experimental (b) Corresponding CFD pressure contours

IV.1. Experimental and CFD Flow Field Analysis

The flow field for experimental and CFD results with two single, and one dual mode operations have been discussed. In dual operation mode, the settling chamber pressure for central jet nozzle (M_{exit} = 4.0) is 395 kPa and surrounding jet nozzle (M_{exit} = 2.68) is 195 kPa. The isentropic exit pressure for this operating condition is 2.60

kPa for central nozzle and 8.64 kPa, however the environmental static pressure in hypersonic test-section is 220 Pa. It can be said that both the jets are underexpanded individually, with the central one having less expansion level than the surrounding one. During the experimental single operation mode for central jet, the settling pressure is 401 kPa and the isentropic exit pressure is 2.64 kPa. In surrounding jet single operating case, the settling chamber pressure is 145 kPa and the corresponding isentropic exit pressure is 6.42 kPa. In both the single operation modes, the exiting jet will be underexpanded. However, the surrounding jet will be exiting from the annular gap nozzle, hence the flow structures will be different for both single operation modes.

Figure 6a shows the schlieren image for the flow around the whole body from the experimental visualization. Figure 6b shows the pressure contours from axisymmetric simulations under the same operating condition. The experimental set-up had mounting structure and connecting pipes as well as pressure tubes connected to the settling chamber, but these are not included in CFD. The front portion of experimental set-up has a double conical nose, which has been fabricated in order to reduce the overall length of the experimental model in limited size. The conical slender body in hypersonic environment has oblique shock wave at the front nose and depending on the length of first conical section and Mach number of flow ($M_{test\text{-}section}$=7 in this study), high pressure region appears at the shoulder of the body and the nose shock wave can take a bow shape as seen in the Fig. 6b. The strength of this nose shock (measure of pressure increase) keeps reducing in downstream direction from the shoulder. It can be seen that nose bow shock does not interact with jet external shock and may not affect the jet-flow field globally because of longer length of slender body. Apart from the nose shock, there is the external jet shock formed near the end of slender body because of expanding jet-flow. However, the environment pressure at the nozzle-exit may be affected by hypersonic flow-field and the length of the slender body hence jet flow field may be only affected by the length of slender body and incoming Mach number. In this study, the hypersonic Mach number remains constant and length of overall body remains the same in experimental study.

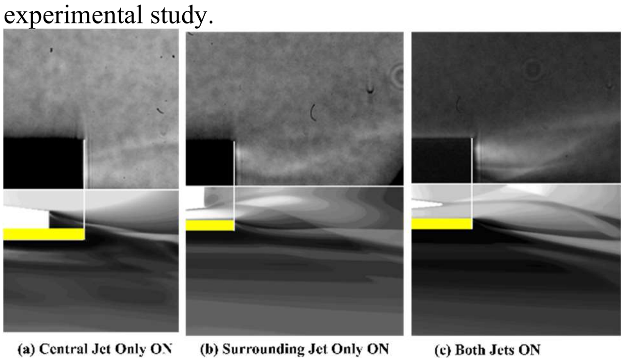


Figure. 7. Jet focused experimental and CFD density contour comparison for (a) Central jet, (b) Surrounding jet and (c) Both jets active.

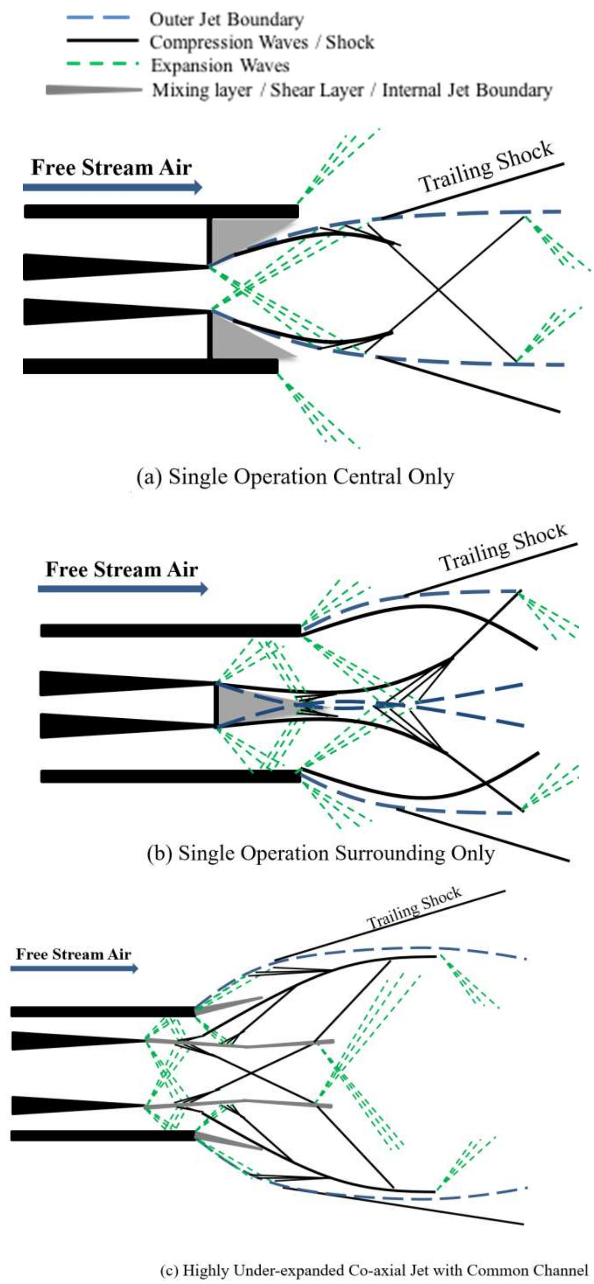


Figure 8. Schematic for shock formations in (a) single central jet operation, (b) single surrounding jet operation (c) Dual jets operations, with common channel

Figure 7 shows the comparison of experimental schlieren (upper half) and numerical density contours (lower half) for previously mentioned two single operation modes: central, surrounding jet and one dual operation mode. Figure 8 shows the corresponding schematic of all three operational modes in experimental model. It should be noted that the experimental model consists of 5.4 mm common channel. The flow field is described in the presence of common channel. It is well known that compression or expansion waves reflect from wall boundary as compression or expansion waves, respectively. However, their reflection from the shear layer or jet boundary is opposite to each other, expansion waves reflect as compression waves and vice-versa. In

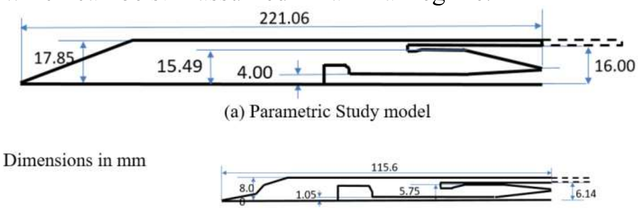
Fig. 7a, the schlieren image and the numerical density contours are compared in single operation central jet case. Although schlieren image only has a faint curved line at jet boundary, which is visible, the CFD simulation shows better flow structures in density contours. In the presence of incoming flow from the freestream, at the corner of slender body, expansion fan forms. This expansion fan tries to turn the incoming flow towards the base; however, the jet exiting from the base tries to expand and deflects the incoming flow away from the centerline, which results in the formation of trailing shock. If there is no incoming freestream flow, the jet may have highly underexpanded and may have formed Mach disk in the first shock cell. The incoming flow from the freestream behaves like an outer jet boundary for the jet exiting from the base. At the central nozzle exit, the expansion fan formed at the nozzle lip, tries to expand the jet. These expansion fans reflect on the jet boundary as compression fronts. These compression fronts interact with each other near the centerline and further reflect from the outer jet boundary as expansion front. Hence, if the common channel length is not enough so that the expansion fans at nozzle lip do not interact with the common channel wall, the central jet in presence of freestream flow may behave like moderately underexpanded jet. Figure 7b shows the schlieren image and the numerical density contours of surrounding jet single operation. The corresponding schematic is shown in Fig. 8b. The surrounding jet single operation mode has slightly more complicated flow structures than the central jet, because the surrounding jet at Mach 2.68 exits from the annular passage. In the presence of incoming freestream expansion fans will be formed at the base corner of the slender body and the formation of trailing shock downstream. The jet exiting from the annular nozzle exit slightly inclined away from the centerline. Initially the expansion fans form at the nozzle inner lip. These expansion fans may interact with the common channel wall and may be reflected as expansion fan. In addition, further downstream flow may interact again with expansion fan formed at common channel exit lip. These multiple interactions with expansion fans may lead to an increase in Mach number for the flow significantly, and these expansion fans may reflect from the jet outer and inner shear layer as compression front. Hence, curved barrel shock and oblique shock may form after the common channel exit. These shocks / compression fronts interact with each other and reflect from jet outer boundary or inner shear layer as expansion front.

Figure 7c compares jet focused schlieren image with density contours of dual operation mode. The corresponding schematic has been shown in Fig. 8c. In presence of dual mode operation of co-axial jet, the flow field has become much more complicated than the single jet flow. There are multiple interactions and reflections of expansion and compression fronts from shear layer, outer jet boundary and common channel wall. The internal flow features of coaxial jets cannot be seen from the experimental schlieren image. However, CFD simulation can give an insight of flow field interaction inside the common channel. Both central and surrounding jets are underexpanded. The expansion fan formed at the nozzle inner lip of central jet reflects from the shear layer between both the central and surrounding nozzle as compression front. These compression fronts form oblique shock, which interact each other near centerline and further, partially reflects from the shear layer as expansion fan and partially transmitted through the shear layer. The expansion fan formed at the inner lip of surrounding jet interacts on the wall and reflects as expansion fan, and it later interacts with the inner shear layer and reflects as compression front. The expansion fronts at the inner lip of common channel exit reflect as compression front from the inner shear layer. These multiple compression fronts can combine and may form stronger oblique shock, which may further reflect from jet outer boundary as expansion fan. However, if there is no common channel, it can be predicted that the expansion fan at the upper lip of central jet nozzle may have reflected at the surrounding jet boundary as weak compression front rather than expansion front in the current case, which may lead to modification of the flow field slightly. This difference in flow field may lead to slight reduction in the length of first shock cell and slight decrease in Mach number in upstream without common channel case. The experimental case had limitation of operating conditions and varying the length of common channel. Hence, further parametric studies have been performed for resized experimental model by varying operating Mach number and common channel length, in order to develop further understanding of jet system performance.

IV.2. Parametric CFD Study for Enlarged (Resized) Experimental Model

In order to understand the effect of common channel on flow-field and different operating Mach number of both jets, the experimental model has been enlarged. In the previous sections, the experimental model has been designed to fit into required area of test section of wind tunnel, which has caused small geometries e.g. throat radius of central jet 2.1 mm, and annular gap of surrounding jet as 0.4 mm. Hence, in order to have reasonable throat dimensions for central and surrounding jets, the throat area of both nozzles for resized model, has been assumed as 5.025×10^{-5} m^2, which corresponds to central jet throat dimeter 8 mm and surrounding jet throat annular gap as 0.5 mm. These changes have led to the increase in common channel internal diameter to 32 mm of overall resized model. The thickness of outer lip has been kept the same as the experimental model (1.85 mm).

The schematic of the resized model for parametric study is shown in Fig. 9a, while is compared with experimental model on same scale in Fig. 9b. Because the front shock does not affect the jet at the rear flow; the double cone nose also has been simplified as conical nose. In order to perform study on the effect of various Mach number, only divergent zone of both the nozzle have been modified in order to generate central and surrounding different Mach numbers while keeping common channel internal diameter same. The Reynolds number of the resized experimental model has changed to 1.6×10^5, which can be still assumed in laminar regime.



(a) Parametric Study model

Dimensions in mm

(b) Experimental model

Figure 9 (a) Enlarged experimental model (b) Actual experimental model in same scale

IV.2.1 Effect of Mach Number with 1-D Isentropic Calculations and CFD With No Common Channel Model

In this section, 1-D isentropic performance calculations are performed on co-axial supersonic nozzle system. The settling pressure for central nozzle is kept at 400 kPa and surrounding jet nozzle is kept at 400 kPa for all the calculation. The jet from these nozzles is exiting at 230 Pa environment pressure without any common channel and having constant exit area shared by central and surrounding nozzle system. The specifications of 1-D calculations are mentioned in Table 4.

Table 4. Isentropic calculation conditions

Parameter	Specifications
Outer Diameter of Body	40.2 mm
Throat Area: Fixed	Central & Surrounding: 5.025×10^{-5} m^2
M_{exit}: varies	Central Jet: 1 - 4.323, Surrounding Jet: vary accordingly
Total Pressure: Fixed	Central Jet: 400 kPa Surrounding Jet: 400 kPa
Total Temperature: Fixed	300 K
Thrust: Calculated	Thrust $= \dot{m}V_{exit} + (P_{exit} - P_a)A_{exit}$

The throat area has been fixed in both the central and the surrounding nozzles. As the central jet Mach number increases, the exit area for central nozzle increases under fixed throat area assumption. This leads to a decrease in the exit area of surrounding nozzle, since the total exit area is constant. Hence, exit Mach number for surrounding nozzle decreases when the throat area is fixed. The variation in surrounding jet Mach numbers has been plotted against central jet Mach number as shown in Fig. 10a. The surrounding jet exit Mach number decreases rapidly to 1, as central jet Mach number increase from 3 onwards.

Further, based on 1-D calculations, the thrust has been calculated for central nozzle, surrounding nozzle and total thrust by adding the individual thrust for the co-axial nozzle. The thrust values are plotted in Fig. 10b against the central jet Mach number. The central nozzle thrust increases linearly as the central jet Mach number increases, while the surrounding jet thrust decreases slowly at first, while decreases rapidly near Mach 4. Accordingly, the total thrust (sum of individual thrust of both nozzles) from 1-D calculation increases slowly in the beginning with increase in central Mach number and dips after Mach 4.

In order to evaluate CFD results based on 1-D isentropic cases, CFD simulations are performed for resized coaxial system in the flow field same as experimental model in Mach 7 freestream for same settling chamber pressure as 1-D isentropic calculations of four different central jet design exit Mach numbers. Accordingly, the surrounding jet design exit Mach number

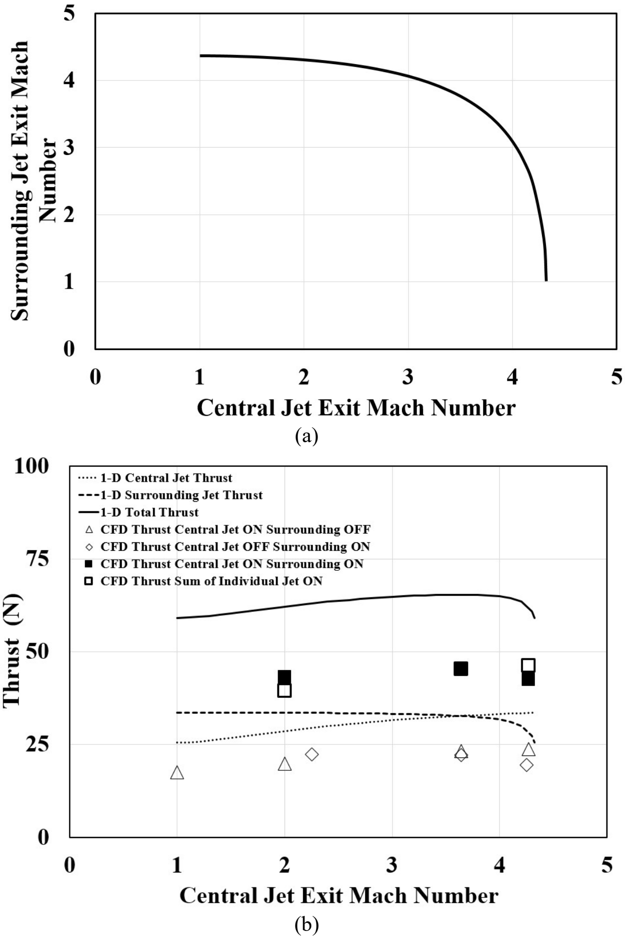


Figure 10 (a) Central and surrounding jet Mach numbers relation (b) Computed thrust from 1-D and CFD calculations (central and surrounding nozzle settling chamber pressures: 400 kPa / 400 kPa)

changes and the nozzle geometries also change in these four different CFD simulations. The simulations have been performed for only central jet active and only surrounding jet active as well as both jets are active. The calculated thrust from CFD is plotted in Fig 10b with data points. When only the central jet is active, the thrust from the system increases linearly as central jet exit Mach number increases. However, thrust values remain below 1-D calculations, because of various interaction losses, which have not been considered in 1-D calculation. Further, when only surrounding jet is active and central jet is inactive, the losses seem to be higher than the 1-D calculation of surrounding nozzle thrust calculations. When both the jets are active, there is a slight gain in thrust than the sum of thrust with individual jet active, up to central jet exit Mach number 3.64 (at this central jet Mach number, the surrounding jet Mach number is almost close. Since the throat area is same for both nozzles, the exit areas of both the nozzles will be same, which means that the total exit area is divided in half). As Mach number increases further, there is a significant decline in the thrust for both jets are active and total thrust is less than the sum of individual thrust, when single jets are active. When comparing the CFD results with isentropic calculations, the trend of CFD thrust is similar to 1-D calculations. However, because of the effects of various losses, there is approximately a 30 % reduction in the total thrust calculated by axisymmetric CFD for the current set-up than the 1-D calculations.

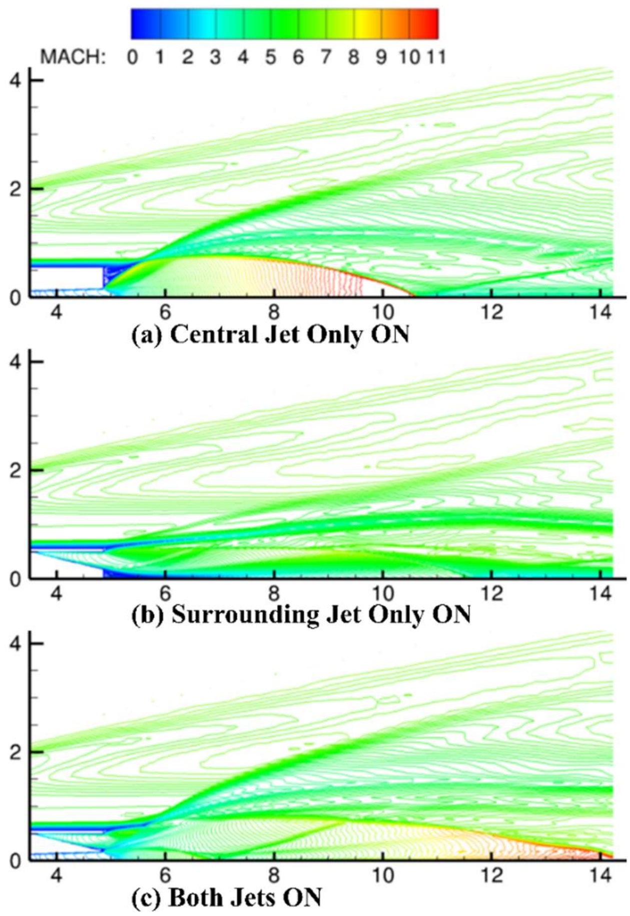


Figure 11 Mach contours for various operation modes corresponding to central jet Mach number 2.0 corresponding surrounding jet Mach number 4.27 and central and surrounding nozzle settling chamber pressures: 400 kPa / 400 kPa) without common channel

In order to understand the flow field of various operation modes without common channel, Fig. 11 shows Mach contours of central jet single operation and surrounding jet single operation as well as both jets dual operation according to Mach 2 central jet nozzle and corresponding Mach 4.27 surrounding jet nozzle. During single operation modes, either central or surrounding ON, the nozzle exit has been assumed closed for the other nozzle. When only central jet is operating (in Fig. 11a), the expansion fan forms at the lip of nozzle exit. It is highly underexpanded and Mach number increases up to 5 just after the nozzle exit and more than Mach 11 at axial location X/D = 10.5. These expansion waves reflected from the jet boundary as compression front, form strong oblique shock, which reduces the Mach number upstream. However, there is no formation of Mach disc in this case.

For surrounding jet operation (at Mach 4.27, in Fig. 11b), the expansion fans form at both the nozzle corner and reflect from the jet boundary near the centerline and outer jet boundary, which causes formation of compression fronts. These two compression fronts interact near axial location X/D = 6.5 and later upstream, are reflected from jet boundaries again as expansion waves. The maximum Mach number in shock cell, before compression front has been Mach 7.5. In dual operating mode case (Fig. 12c), when the central and the surrounding jets are active at Mach 2 and 4.27, respectively, the central jet could not expand highly as in case of only central jet operation (Fig. 12a) in presence of surrounding jet and also surrounding jet does turn away than only surrounding jet operation. The overall length of the combined shock cell of the jet in dual mode operation increases up to X/D = 14.5 in comparison to only central jet operation. However, the shock cell consists of small shock-cell of central jet, which forms up to X/D = 6.5. The multiple interactions of expansion and compression front occur in dual mode and maximum Mach number reaches up to 10.7, which is lower than central jet operation. The main conclusion from this section is that the flow field have been significantly altered in dual mode operation and shock cell length increases in presence of surrounding jet, however in the central jet Mach 2 case, the thrust in dual mode operation is higher than sum of individual single jet thrusts without common channel.

IV.2.2 Effect of Common Channel on Thrust Performance

In this section, the effect of common channel length on thrust performance of co-axial jet system in two single and one dual operation mode is analyzed using axisymmetric CFD simulations. Figure 12 shows the effect of central jet exit Mach number with various common channel lengths by plotting thrust. The primary X-axis shows the central jet Mach number and secondary X-axis shows corresponding surrounding jet Mach numbers. In case of single central jet operation, the surrounding jet Mach number is zero, and in case of surrounding jet only Mach number, the central jet is closed. The common channel length of 29.5 mm from the exit of central jet nozzle is called short common channel, while the common channel of length 59 mm from the central jet nozzle exit is called long common channel. In these plots, the common channel length zero is the same as isentropic calculations. The operating settling chamber pressure for single or dual mode operations are 400 kPa each.

When only the central jet is operating (Fig. 12a), at Mach 1, 3.64, and 4.27 there is no significant effect of short and long common channel length on total thrust. The thrust of the system increases with increase in Mach

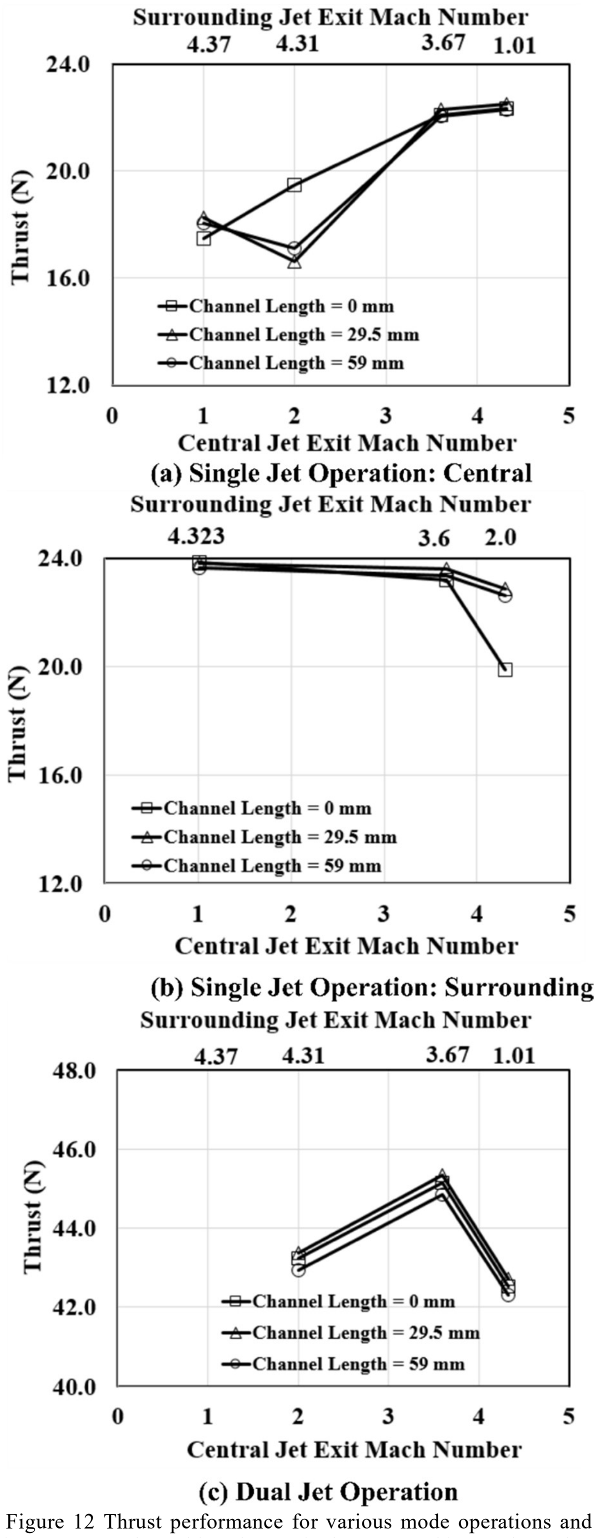


Figure 12 Thrust performance for various mode operations and various common channel length. X-axis shows central jet Mach number and corresponding surrounding jet Mach number according to Fig. 10a

number without common channel from approximately 18 N to 22.3 N. However, at Mach 2, there are approximately 14.6 % and 12.2 % decreases in performance of single jet system in presence of short or long common channels, respectively. The reasons for this loss in performance are explained in later discussion, when Mach contours are compared. In Fig. 12b, the single operation mode for only surrounding jet operation thrust have been plotted. At high surrounding jet Mach numbers, 4.323 and 3.6, in single operation mode, there is no significant effect of common channel, while for lower surrounding jet Mach number 2.0, there are approximately 15 % and 17.4 % increases in thrust in presence of short and long common channels, respectively. This behavior is opposite to the central jet only operation at Mach 2. Figure 12c shows the effect of common channel length for dual mode operation at various Mach numbers. It can be seen that there is a slight increase in thrust under short common channel length and there is a slight decrease in thrust from long common channel in dual operation modes. However, the increase and decrease in thrust for short and long common channels, respectively, are less than 1%. Hence, it can be concluded that in dual operation mode, the presence of common channel does not affect the thrust performance of the system. Further discussions about the difference in these performances due to common channel are performed in the next section with the flow-field analysis.

IV.2.3 Effect of common channel in Single Operation Mode: Central Jet at Mach 2.0

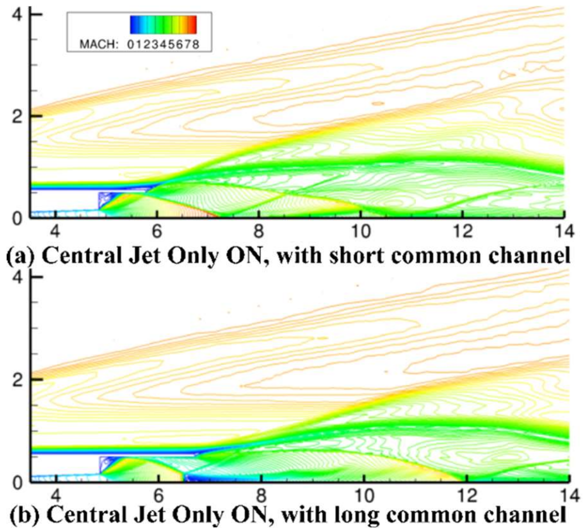
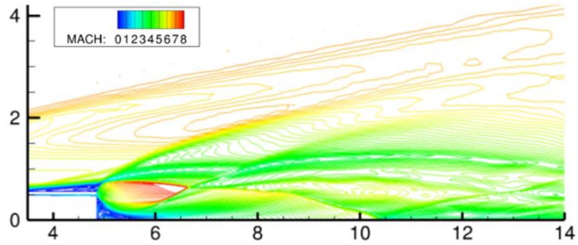


Figure 13 Mach contours for central jet single operation at Mach 2 with (a) common channel length 29.5 and (b) common channel length 59 mm (The corresponding without common channel Mach contours is shown in Fig. 11a)

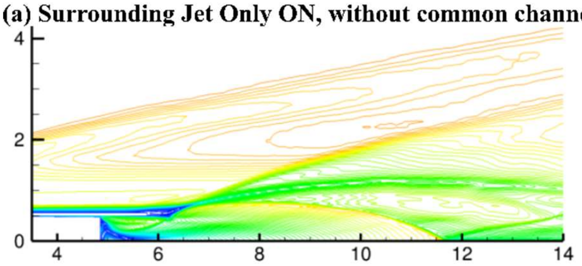
Without common channel (Fig. 11a), there are expansion fans originating from the nozzle lip, which reflect from the jet shear layer as compression front and these compression front forms oblique shock wave, which may interact with the oblique shock from the other end near the centerline at X/D = 10.5. However; in presence of common channel with single central jet operation (Fig. 13a), the expansion fans hit the common channel wall and reflected as expansion front, which increases the Mach number so high that it is required to form a compression front / shock wave inside the common channel, to turn further the flow away from the centerline. The formation of compression front inside the common channel, before exiting from the common channel, leads to loss in thrust in the presence of common channel, for central jet single mode operation at Mach 2. For longer common channel (Fig. 13b), the interaction point of expansion fans is well inside the channel, and the flow remained straight along the wall, however the formation of small Mach disk near the center, leads to thrust loss in presence of longer common channel. The thrust loss is slightly less for longer common channel, because the flow is less expanded near the common channel exit in case of longer channel than short channel. The alteration of flow field with common channel in central jet single operation and formation of compression front before the common channel exit, leads to thrust loss of the system.

IV.2.4 Effect of common channel in Single Operation Mode: Surrounding Jet at Mach 2.0

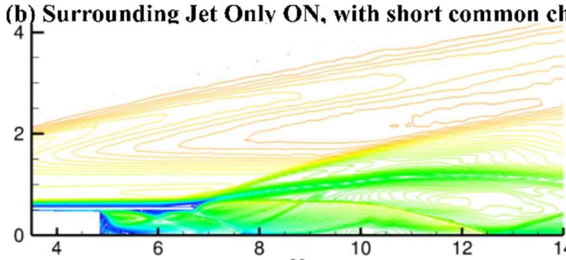
The surrounding jet single operation mode has different geometric and flow topology than the central jet single operation. The surrounding jet has nozzle exit as annular passage and the inner region at the exit of annular passage may have less pressure, which can lead to high expansion of surrounding jet towards the centerline as well as near the outer region at the exit of annular passage. Figure 14 shows the Mach contours of surrounding jet single mode operation for exit Mach number 2. Figure 14a shows Mach contours from CFD simulations without any common channel. The Mach contours for short and long common channels are shown in Figs. 14b and 14c, respectively with surrounding jet single operation. From the previous section about thrust performance, it has been revealed that Mach 2 surrounding jet has positive effect in thrust performance with common channel. There is a 15 % and 17.4 % increment in thrust with short and long common channels, respectively in comparison to no common channel case. The reason for this behavior can be explained by Mach contours. From Mach contour without common channel (Fig 14a), it can be seen that the exiting annular jet expands towards the centerline near the inner corner of annular passage flowing away from the centerline at the outer corner of annular exit. It causes shock-cell to be slightly tilted away from the centerline, subsequently turning the flow away from the jet axis, which requires compression fronts / shockwaves to turn the flow again towards centerline. In presence of short common channel (Fig. 14b), the channel wall prevents high expansion at the outer lip of annular passage and wall making flow straight. Moreover, the expansion fan formed at the nozzle inner lip may be reflected at the wall as expansion fans, which increases exit Mach number more than 2. This leads to increase in thrust performance in single operation mode with surrounding jet. With longer common channel (Fig. 14c), though, the compression front inside the channel interacts with the wall, causing flow separation on the channel wall. This interaction at shear layer may lead to expansion front. This again increases the Mach number at the exit of common channel, and it causes better performance than the short channel.



(a) Surrounding Jet Only ON, without common channel



(b) Surrounding Jet Only ON, with short common channel



(c) Surrounding Jet Only ON, with long common channel

Figure 14 Mach Contours for single mode surrounding jet operation at Mach 2 with (a) no common channel (b) common channel length 29.5 and (c) common channel length 59 mm

It can be concluded from the single operation mode that common channel works positively with surrounding annular jet leading to an increase in the thrust, while affecting negatively for central jet because of significant alteration in flow field with short or long common channel. It should be noted that when central jet has been designed at Mach 2, the corresponding surrounding jet Mach number was 4.31 not 2. From the thrust performance discussion about the effect of common channel, it has been found out that Mach 2 central and Mach 2 surrounding nozzle in single operation mode have thrust close to each other, approximately 19 N. Although the exit jet topology is different in surrounding and central jet single mode operation. In the presence of common channel, with Mach 2 central jet, the computed thrust reduces, while with Mach 2 surrounding jet, the computed thrust increases.

IV.2.5 Effect of common channel on Thrust in Dual Operation Mode: Central Jet Mach 2

In this section, the Mach contours of dual mode with and without common channel are discussed under central jet Mach number 2 and corresponding surrounding jet Mach number 4.31. Figure 15 shows the Mach contours of dual mode with short and long common channels. The corresponding Mach contour without common channel case is shown in Fig. 11c. It can be seen that with surrounding jet, the under expansion of central jet is slightly reduced and the shock length is slightly increased

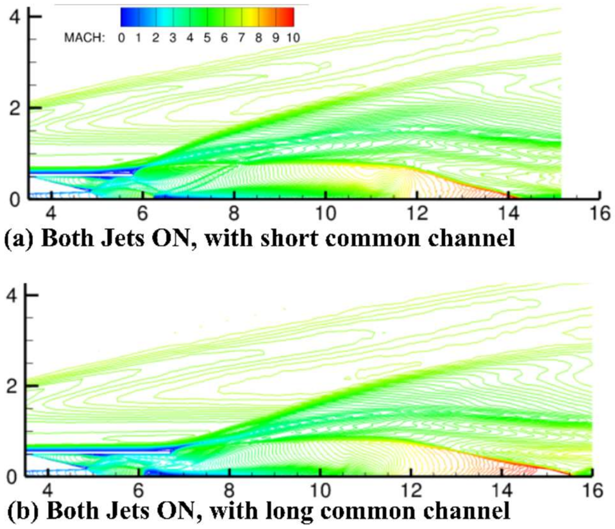


Figure 15 Mach Contours for dual mode operation at central jet Mach 2 with (a) common channel length 29.5 mm and (b) common channel length 59 mm (The corresponding without common channel Mach contours is shown in Fig. 11c)

in dual mode, in comparison to central jet single operation. Without common channel, the shock-cell of surrounding jet tilted away from centerline. Fig. 15a shows that in the presence of short common channel, the shock cell of surrounding jet is significantly modified because of interaction with the common channel wall. The double barrel shock has been formed near the centerline. A small Mach disk is formed near the jet axis after the common channel exit. However, the flow field near the nozzle exit, has been significantly modified by common channel, but there is no significant effect on total thrust. It may be because, the contribution from central jet near the common channel exit may have improved, but because of flow separation near the wall, the surrounding jet thrust may be slightly reduced, which lead to little effect on thrust with small common channel. In Fig. 15c, with longer common channel, a larger Mach disk, inside the common channel has been formed near the jet axis, compared to short common channel. With short common channel, the shock cell length has been approximately X/D = 14, hardly changed compared to no common channel case. In presence of longer common channel in dual mode, the shock cell became larger up to X/D = 16.0. Although there is larger Mach disk near the centerline with long common channel, there is slight reduction in thrust with long common channel in dual mode, but the difference is not more than 1%. Flow field is altered significantly in presence of long common channel.

V. Conclusion

The main motivation of this study has been to evaluate the performance and the flow-field of two coaxial jet systems operating in hypersonic environment with the addition of a common channel to the coaxial exhaust flow nozzle. Initially the experimental studies have been conducted for small slender body having two high-pressure chambers and having central and surrounding nozzle with common channel, which exhausted the supersonic flow in hypersonic environment. Although the experiments model can only provide qualitative schlieren images, the numerical simulations are also conducted in order to evaluate the performance and to compare flow field with experiments in order to validate CFD results qualitatively. Single point pressure measurement on the wall of common channel has been also performed to compare with numerically computed pressure in various single and dual operation modes. Further, parametric study based on 1-D isentropic calculations and CFD simulations have been performed for slightly enlarged slender body with two co-axial supersonic jets without common channel to understand the effect of varying Mach numbers of central and surrounding jets. The main findings in case of no common channel by 1-D and CFD studies is that the central and surrounding should have same total allowable exhaust area at dual jets operating mode in order to achieve higher total thrust than sum of thrust from individual jets operating in single operation mode.

The introduction of short and long common channel in single and dual operation modes have significantly modified the jet flow-field but the main advantage in performance is only observed in single mode surrounding jet operation. At Mach 2 surrounding jet single mode operation, the thrust has increased 15-17.4 % with short and long common channel than no common channel case. However, in central jet single mode operation, the thrust has decreased by 12.2-14.6 % in presence of short and long common channel. These differences in performances of single operation modes are because of difference in flow structures in presence of common channel for conical jet (in central jet only operation) and annular jet (in surrounding jet operation). The presence of common channel has a negligible effect on thrust performance when operating dual jets together. However, it can also be noted that the best distribution of exit area is when both jets central and surrounding have the same or similar injection Mach numbers exhausted into the common channel.

Acknowledgements

The author acknowledges the Japanese Government, Ministry of Education, Culture, Sports, Science, and Technology, Monbukagakusho (MEXT) scholarship program for providing support for the completion of this research.

References

[1] Peter W. Merlin: Design and Development of the Blackbird: Challenges and Lessons Learned, AIAA 2009-1522, *47th AIAA Aerospace Sciences Meeting including The New Horizons Forum and Aerospace Exposition, Orlando, Florida, 05 January 2009 - 08 January 2009.*

[2] U.B. Mehta, M.J. Aftosmis, J.V. Bowles, S.A. Pandaya, Skylon Aerodynamics and SABRE Plumes, AIAA-2015-3605, *20th AIAA International Space Planes and Hypersonic Systems and Technologies Conference, Glasgow, Scotland, 6-9 July 2015.*

[3] E. Rathakrishnan, Applied Gas Dynamics, second ed. (John Wiley & Sons NJ., USA, 2019, pp 505-509).

[4] H. Sharma, A. Vashishtha, E. Rathakrishnan, P. Lovaraju, Experimental study of overexpanded co-flowing jets, *The Aeronautical Journal 112(1135), 2008, 537-546.* doi:10.1017/S0001924000002499

[5] R.M. Murli, S.M.A. Kumar, E. Rathakrishnan, Characteristics of Parallel and Inward Canted Sonic Twinjets, *International Review of Aerospace Engineering (IREASE), Vol.11, No. 4 (2018), pp 138-145.*

[6] S.N. Belkacem, A. Beghifja, Numerical Investigation of Coaxial Turbulent Jet, *International Review of Mechanical Engineering (IREME), Vol 13, No 2 (2019) pp 78-86*.

[7] T.A. Edwards, Numerical investigation of hypersonic exhaust plume/afterbody flow fields, *10th Australasian Fluid Mechanics Conference, Proc. Vol. 2, 1989, pp 15.29-15.32.*

[8] B.W. Oudheusden, F. Scarano, PIV Investigation of Supersonic Base-Flow–Plume Interaction, *Topics in Applied Physics Volume 112, 2008, pp 449-458.*

[9] S. Stephan, R. Radespiel, Propulsive jet Simulation in a Hypersonic Ludwieg Tunnel, *DLR Report, 2012, Document ID: 28313.*

[10] C.W. Clifton A.D. Cutler, A supersonic Argon/Air Coaxial Jet Experiment for Computational Fluid Dynamics Code Validation, *NASA Technical Report, NASA/CR-2007-214866*.

[11] R.A. Baurle, J.R. Edwards, Hybrid Reynolds-Averaged / Large Eddy Simulations of a Co-axial Supersonic Free-Jet Experiments, AIAA-2009-129, *47th AIAA Aerospace Sciences Meeting including The New Horizons Forum and Aerospace Exposition, Orlando, Florida, 05 January 2009 - 08 January 2009.*

[12] K.B.M.Q. Zaman, I.M. Milanovic, A.F. Fagan, C.J. Miller, Experimental and computational study of tones occurring with a co-axial nozzle, *International Journal of Aeroacoustics, Vol 18, issue 2-3, 2019, pp. 189-206.*

[13] S.E. Cliff, M.F.Denison, S.M. Yekta, D.E. Morr, D.A. Durston, Wind Tunnel Model Design for Sonic Boom Studies of Nozzle Jet with Shock Interaction, AIAA-2016-2035, *54thAIAA Aerospace Science Meeting, Sci-tech, January 2016, California, USA.*

[14] V. S. Ivanov, S. M. Frolov, V. I. Zvegintsev V. S. Aksenov , I. O. Shamshin , D. A. Vnuchkov, D. G. Nalivaichenko , A. A. Berlin, V. M. Fomin, A. N. Shiplyuk and N. N. Yakovlev: Hydrogen-Fueled Detonation Ramjet Model: Wind Tunnel Tests, *AIP Conference Proceedings, 2027, 030041 (2018).*

[15] P.V. Bulat, O.N.Zasuhin, and V.N. Uskov, On classification of flow regimes in a channel with sudden expansion. *Thermophysics and Aeromechanics, 2012, vol. 19, No. 2. pp 233-246.*

[16] G. C. Layek, C. Midya, S. Mukhopadhyay: Effects of Suction and Blowing on Flow Separation in a Symmetric Sudden Expanded Channel. *Nonlinear Analysis: Modelling and Control, 2008, Vol.13, No.4, pp 451-465.*

[17] O. Imamura, T. Watanuki, K. Suzuki, Kashiwa Wind Tunnel Working Group (Univ. of Tokyo), Flow characteristics of UT-Kashiwa Hypersonic Wind Tunnel, *Proceedings of 39th Fluid Dynamics Conference / Aerospace, Numerical Simulation Symposium 2007, JAXASP-07-016, pp 50-55.*

[18] M. Martini, A. Smoraldi, L. Cutrone, M. Zanchetta, R. Varvill, Analysis of Lapcat A2 Vehicle Scimitar Engine Nozzle, *XXII Conference of Italian Association of Aeronautics and Astronautics, Napoli 9-12 September 2013*.

[19] K. E. Tatum, L D. Huebner, Exhaust Gas Modelling Effects on Hypersonic Powered Simulation at Mach 10, AIAA-95-6068, *International Aerospace Planes and Hypersonic Technologies, 1995, TN, USA*

[20] M.S. Liou, A sequel to ASUM, Part II: ASUM+ -up for all speeds", *Journal of Computational Physics, Vol.214, No.1, 2006, pp 137-170.*

[21] C. Hirsch, Numerical Computation of Internal and External Flows Vol.1: Fundamental of Computational Fluid Dynamics, *second ed. (John Wiley and Sons, 2007, Singapore).*

Authors' information

[1] Doctoral Student, Department of Advanced Energy, The University of Tokyo, 5-1-5 Kashiwanoha, Chiba, Japan

[2]Assistant Lecturer, Department of Aerospace, Mechanical and Electronic Engineering, Institute of Technology, Carlow, Ireland.

[3]Assistant Professor, Department of Aeronautics and Astronautics, Graduate School of Engineering, The University of Tokyo, 7-3-1 Hongo, Bunkyo-ku, Tokyo, Japan.

[4]Professor, Department of Advanced Energy, The University of Tokyo, 5-1-5 Kashiwanoha, Chiba, Japan

Mr. Mohammad Samara was born in Madinah, Saudi-Arabia in 1989. He graduated As B.S in aeronautical engineering from Jordan University of Science and Technology in 2013, and received his Masters from the University of Tokyo/ Advance energy department in 2015, currently he is pursuing his Doctoral Degree from the department of Advanced Energy in the University of Tokyo as he is conducting research related with Flow Field and Performance Analysis of a Common Channel Added to a Coaxial Nozzles for Future Hypersonic Vehicles such research includes both Experimental and Numerical Studies.

Mohammad Samara has a 4 years industrial experience in the field of Aircraft Engine maintenance, Design and manufacturing and CFD analysis Engineer. Currently his is working in Mitsubishi-FUSO as a CFD R&D engineer.

Dr. Ashish Vashishtha was born in Bisalpur, UP, India in 1984. He is currently working as Assistant Lecturer at Aerospace, Mechanical and Electronic Engineering, IT Carlow, Ireland. He had received his PhD from The University of Tokyo Japan in 2016, in the area of experimental and numerical unsteady hypersonic flows. He also had 4.75 years of industrial experience regarding combustion modelling in IC Engines. He completed his bachelor and master's degrees in Aerospace Engineering at Indian Institute of Technology Kanpur, India in 2008. His current research interests are shockwave applications in various transdisciplinary industry oriented applied fields as well as combustion and detonation modelling. He is currently member of the American Institute of Aeronautics and Astronautics (AIAA) and the American Society of Mechanical Engineers (ASME).

Dr. Yasumasa Watanabe was born in Japan in 1987. He graduated from Department of Aeronautics and Astronautics, the University of Tokyo and received his B.S. in 2009 and M.S. in 2011. He received his Ph.D. in Aerospace Engineering from the University of Tokyo in 2014. The major field of study is hypersonic aerodynamics and flow actuation by electrical discharges. His current research interests are experimental plasma aerodynamics, diagnostics of low-temperature plasma and hypersonic aerothermodynamics.Dr. Watanabe is currently a member of the Japan Society for Aeronautical and Space Sciences (JSASS), the American Institute of Aeronautics and Astronautics (AIAA), and American Physical Society (APS).

Professor Kojiro Suzuki was born in Tokyo, Japan in 1962. He was graduated from Department of Aeronautics and Astronautics, Graduate School of Engineering, the University of Tokyo and has gotten the doctor degree in Engineering from the University of Tokyo, Tokyo, Japan in 1990. The major field of study is the hypersonic aerodynamics and aerothermodynamics in relation to the space transportation system and the planetary exploration. The current research interests are the thermochemical nonequilibrium flow around hypersonic bodies, the deployable membrane aeroshell for atmospheric entry vehicles, and small planetary landing probes. Prof. Suzuki is a member of the Japan Society for Aeronautical and Space Sciences (JSASS) and the American Institute of Aeronautics and Astronautics (AIAA).