Doctoral Thesis (Abridged)
博士論文 (要約)

Flow Field and Performance Analysis of Common Channel

Added to

Coaxial Nozzles for Future Hypersonic Vehicles

(将来の極超音速機用の同軸ノズルに付加された共通ダクトの

流れ場と性能解析)

Samara　Mohammad　Saeed

サマラ　ムハマド　サイード

Doctoral Thesis (Abridged)

博士論文 (要約)

Flow Field and Performance Analysis of Common Channel Added to Coaxial Nozzles for Future Hypersonic Vehicles

（将来の極超音速機用の同軸ノズルに付加された共通ダクトの流れ場と性能解析）

Samara　Mohammad　Saeed

サマラ　ムハマド　サイード

Department of Advanced Energy,
Division of Transdisciplinary Sciences,
Graduate School of Frontier Sciences,
The University of Tokyo, Kashiwa

January, 2020

Dedication

This work is dedicated to my mother Mrs. Arwa Bustami, and father Dr. Saeed Samara, my late Grandmother Mrs. Fathia Bustami, my friend and teacher Dr. Ashish Vashishtha, my brother Mr. Ahmad Samara, and his Wife Mrs. Manal Abd-alghany to my sister Mrs. Rua Samara and her husband Mr. Moutasem Shahin, to my other sister Miss. Rawand Samara and to my friend and brother Mr. Anas Samara.

Abstract

Flow Field and Performance Analysis of Common Channel Added to Coaxial Nozzles for Future Hypersonic Vehicles

将来の極超音速機用の同軸ノズルに付加された共通ダクトの流れ場と性能解析

Samara　Mohammad　Saeed
サマラ　ムハマド　サイード

Doctoral Thesis
Graduate School of Frontier Sciences, the University of Tokyo
Division of Transdisciplinary Sciences, Department of Advanced Energy

Supervisor: Professor Kojiro Suzuki

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 co-axial engine configurations. Such configuration had been applied historically in the SR71 black bird engines, which integrates the Pratt and Whitney J58 engine with a ramjet engine. Co-axial configuration was also proposed for the hypersonic air breathing flight as the Synergistic Air-Breathing Rocket Engine (SABRE). In the SABRE engine configuration, an air-breathing rocket engine is combined with ramjet as an axisymmetric co-axial configuration. However, there is limited fundamental understanding of single or dual mode operations of combined co-axial nozzles in hypersonic environment and its impact on thrust performance and what will be the effect of adding a common channel that have the potential of augmenting the thrust of the coaxial jets sharing a common total external area. The current study focuses on

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 through a proposed common channel. in order to study the augmentation of total thrust from both the nozzles, the wall of outer nozzle have been extended so that central and surrounding jets interact in a confined passage before exhausted into the low-pressure hypersonic external flow. In this research, this confined passage is termed as common channel. Single and dual operation modes for co-axial nozzles with common channel can be descried as; 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 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 length of 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 expansion level and Mach disk. 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.

The main objective of current study is to evaluate the performance of co-axial nozzle in hypersonic environment and to study the effect of the common channel to enable thrust augmentation for various operation modes. The study is conducted by qualitative experimental flow visualization of shock structure for single and dual operation modes with common channel. Further, numerical simulations are performed for cases to evaluate the performance of system by thrust calculations. Further parametric study has been performed for resized model by isentropic as well as CFD simulations for various central and surrounding jet Mach numbers without common channel. Further, CFD simulations have been performed to study the effect of common channel on flow field and thrust performance.

Coaxial engines exhaust jets flow has an impeded complexity in regards how to model the exhaust jet flows as well as the flow structure within and outside the body, not to mention the effect of using a cold airflow in the experiment to simulate the actual hot exhaust engines jet flow; In the current research the experimental model is an axisymmetric slender body having a jet generator in it, is consider to simulate a combined cycle engine for the hypersonic flight. The jet generator is composed of two coaxial axisymmetric nozzles and a short common channel, in which two jets meet before being exhausted at the base of the body. The experiments have been performed for small slender body kept in hypersonic Mach 7 flow, which consists of two 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, called 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 were validated with experimental images. Later, the parametric study has been performed 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, that is Single central jet operating only, surrounding jet operation only, and dual operational mode in which both jets are in operation.

The main motivation of current study was to evaluate performance and flow-field of two coaxial jet system operating in hypersonic environment with the addition of a common channel to the coaxial exhaust flow nozzle. Initially the experimental study have been conducted for small slender body having two high-pressure chamber 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 to evaluate performance and compare flow field with experiments in order to validate CFD results qualitatively. Single point pressure measurement on the wall of common channel was 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 have increased 15-17.4 % with short and long common channel than no common channel case. However, in central jet single mode operation, the thrust have 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 have 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.

Suddenly expanded nozzle is well studied for the case of single jet expanding in higher area channel. However, the effect of surrounding jet on a suddenly expanded central jet in hypersonic environment has not been studied in the literature according to the knowledge of authors. The current study will evaluate the performance of common channel in following cases: single operation mode with central jet expanding in the common channel, surrounding jet expanding in the common channel, and the effect of the common channel when both jets injected in the common channel. The main objective of current study is to evaluate the performance of co-axial nozzle in hypersonic environment and study the effect of the common channel to enable thrust augmentation for various operation modes. The study is conducted by qualitative experimental flow visualization of shock structure for single and dual operation modes with common channel. Further, numerical simulations are performed for cases to evaluate the performance of system by thrust calculations. Further parametric study has been performed for resized model by isentropic as well as CFD simulations for various central and surrounding jet Mach numbers without common channel. Further, CFD simulations have been performed to study the effect of common channel on flow field and thrust performance.

As per the current research the use of the common channel is recommended for a coaxial jet with a surrounding jet that have a relatively low exit Mach number (less than 3 Mach at the nozzle exit into the common channel) with under-expanded jet condition, on the other hand; when having such system of coaxial jets sharing the same exhaust exit area; most of the common area should be utilized by the central jet nozzle; in such descried configuration the common channel will be effective during which the surrounding jet only is in operation. A possible scenario of an effective use of the common channel is when having a coaxial configured RBCC (Rocket Based Combined Cycle) in which the common exit area will mainly used by a central rocket engine that will be active during the subsonic and in space flight regime, and during the hypersonic regime the surrounding jet (can be set as a scram jet) only will be in operation; in which by using the common channel the expansion of the exhaust flow will be improved resulting in an increase in thrust when operating with the use of the surrounding jet only.

For the future work, it is recommended to study the process of coaxial jets flows starting; such starting process can have an interesting behavior of jets as it get through from starting phase to steady state phase of operation; it is interesting to investigate the starting sequence of the coaxial jets system and how it will affect the flow structure during the stating phase. For the common channel, it is also interesting to study the effect of the common channel as it operates through different free stream Mach numbers from subsonic, supersonic to hypersonic flow; and the effect of the free stream pressure on flow structure and performance of the coaxial jets system common channel.

Acknowledgments

I would like to express my gratitude first to my parents for their support and respect to my decision in pursuing aerospace engineering, as my major and future carrier, along with my parents I would like to acknowledge all the people that believed in my thorough my journey and give me a truthful guidance and support, special thanks for my family members my brothers and sisters Anas, Ahmad, Rua, and Rawan. Along with my Friends around the world.

I would like to thank my professor; Prof. Kojiro Suzuki; for accepting me into his well-known research laboratory, and for his patience in guiding and supporting me in learning the very fundamental science and know how behind CFD, and applying it to applications that fits my interest to the fascinating field of hypersonic flight using computational engineering.To my professor's assistant; Dr. Yasumasa Watanabe; for being a continuous source of knowledge, and being a row model for detection and hard work, along with providing well appreciated help and support in both CFD, and Experiments. And to my Friend and teacher Dr. Ashish Vashishtha, for being a second adviser to me and through our discussions I got to learn a huge amount of knowledge.

Additional thanks to the scientists and engineers of JAXA (Japanese Aerospace Exploration Agency) for providing me with a well appreciated advices and insight on the Aerospace research methodologies. Finally I would like to acknowledge the Japanese Government, Ministry of Education, Culture, Sports, Science, and Technology (MEXT) scholarship program for providing me support for my Master’s program, and giving me the chance to come and study in the great country of Japan.

Table of Contents

List of Figures

A.2.5	300 K, jet stagnation temperature, effect on jet flow axial velocity contour
A.2.6	1000 K, jet stagnation temperature, effect on jet flow axial velocity contour
A.2.7	3000 K, jet stagnation temperature, effect on jet flow axial velocity contour
A.2.8	300 K, jet stagnation temperature, effect on jet flow density contour
A.2.9	1000 K, jet stagnation temperature, effect on jet flow density contour
A.2.10	3000 K, jet stagnation temperature, effect on jet flow density
A.2.11	300 K, jet stagnation temperature, effect on jet flow pressure contour
A.2.12	1000 K, jet stagnation temperature, effect on jet flow pressure contour
A.2.13	3000 K, jet stagnation temperature, effect on jet flow pressure contour
A.2.14	300 K, jet stagnation temperature, effect on jet flow Mach number contour
A.2.15	1000 K, jet stagnation temperature, effect on jet flow Mach number contour
A.2.16	3000 K, jet stagnation temperature, effect on jet flow Mach number contour

List of Tables

Nomenclature

List of Symbols

c	:	speed of sound
C_p	:	pressure coefficient
L	:	length
M	:	Mach number
P	:	pressure
q	:	dynamic pressure
R	:	specific gas constant for air
Re	:	Reynolds number
t	:	dimensional time
T	:	temperature
h	:	total enthalpy
J	:	Jacobian of transformation
τ	:	shear stress
$\dot{q}$	:	heat flux
u	:	velocity in the x direction
v	:	velocity in the y direction
x	:	x Cartesian coordinate direction
y	:	y Cartesian coordinate direction
γ	:	specific heat ratio for air
μ	:	dynamic viscosity
ρ	:	density
A	:	cross section area
mx	:	number of mesh points in the x direction
my	:	number of mesh points in the y direction
ξ	:	generalized coordinate first direction
η	:	generalized coordinate second direction
n	:	normal
$\dot{m}$	:	mass flux
e	:	internal energy
Pr	:	Prandtl number
L	:	characteristic dimension
k	:	thermal conductivity
y	:	in the y direction
ξ	:	generalized coordinate first direction
η	:	generalized coordinate second direction
r	:	cartesian coordinate position vector magnitude
i	:	unit vector in the direction of x
j	:	unit vector in the direction of y

Subscripts

0	:	total value
∞	:	free stream

v	:	viscous component of flux vector
ref	:	reference
n	:	normal component
x	:	in the x direction
y	:	in the y direction
ξ	:	generalized coordinate first direction
η	:	generalized coordinate second direction
L	:	from the left direction
R	:	from the right direction
$ZoneA$	:	at zone A
$ZoneB$	:	at zone B
$inlet$	:	at the inlet
$wall$	:	at the wall

Superscripts

*	:	non-dimensional quantity
^	:	generalized component of flux vector
n	:	quantity at discrete time level n
+	:	from the left direction
-	:	from the right direction

Chapter 1

Introduction

In order for futuristic hypersonic aircraft to operate efficiently in different speed regimes; subsonic, supersonic, and Hypersonic; engines with combined cycles can be applied by having a single propulsion system that will operate once the aircraft have retched to hypersonic speed, such system will require another propulsion system that will accelerate the aircraft to such speeds, one simple configuration to integrate such propulsion systems is the coaxial configuration. Integration of multiple propulsion systems in a coaxial configuration is one of the challenges to be applied in hypersonic flight regime. The current 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, along with an examination of the effect of adding a common channel to the coaxial jets.

1.1 Background

1.1.1 Sustained Hypersonic Flight Propulsion System:

There are some proposed engines systems to power Hypersonic Aircrafts in order to achieve a relatively long sustained hypersonic flight; such as the scramjet engine [1]; such engine can achieve supersonic combustion and uses the aircraft fore-body to initially compress the air in which will work as an external inlet for the scramjet engine, the engine will receive air through its inlet, following the inlet there will be an Isolator in which will stabilized the air flow entering the engine and cause a farther increase in air pressure; that will be essential for the combustion to take place at the supersonic speeds. The combustion will increase the flow enthalpy, and will cause the flow to increase its pressure; such pressure will be reduced by expanding it through an internal nozzle, adding momentum to the flow and creating thrust, on the other hand and due to the high pressure flow exhausted from the internal nozzle; exiting flow can be further expanded externally, and by having an aft-body ramp of the aircraft some of the flow power can be harvested and turned into more thrust as a result of pressure build up on the aft-body ramp surface. In order to achieve a relatively low speed hypersonic flight a ramjet [2] engine can be used, such engine operates similar to the Scramjet engine however it contain a normal shock wave at its inlet such shock wave will decelerate the flow to subsonic speeds in which the flow will be mixed with fuel and combusted in a subsonic flow environment. Neither scramjet nor ramjet can accelerate the aircraft to the supersonic regime; to achieve that an additional system must be added to work with the mentioned engines, such system can be either a rocket engine or even a turbine based jet engine, as the turbojet or a turbofan. A turbo-ramjet engine [3] is a combined cycles engine that have been tested and operated on the SR71 Black Bird, such aircraft uses the Pratt & Whitney J58 Jet Engine, as the engine core, along with six bypass tubes that will transfer the air from the aircraft inlet towards that engine after burner, all in which will operate as a ramjet, the engine operate differently depending on the Mach number and the thrust percentage generated by etch cycle varies accordingly. Although the SR71 is a supersonic flyer a hypersonic flyer can be developed to combine a scramjet with a turbojet. It is worth mentioning the JAXA's precooled turbojet engine [4] with an afterburner; the mentioned JAXA's engine contain an airflow pre-cooler that will cool dawn the hot inlet flow before it enters the compressor, the pre-cooler uses liquid hydrogen that will also be used as a fuel for the engine, this engine has an after burner which will heat the flow before it is discharged from the engine nozzle. A system that have a coaxial engines configuration is the Skylon SABRE (Synergistic Air-Breathing Rocket Engine) [5] Engine, which can is considered as a form of RBCC (Rocket Based Combined Cycle) engine, such engine combine two main propulsion systems, first is an air breathing rocket engine that will cool dawn the hypersonic free stream air, and liquefy it all in which to be injected in a rocket engine along with the necessary fuel for combustion. The second propulsion system is a ram jet engine that will burn some of the partially cooled flow, the ram jet exhaust flow will surround the rocket engines.

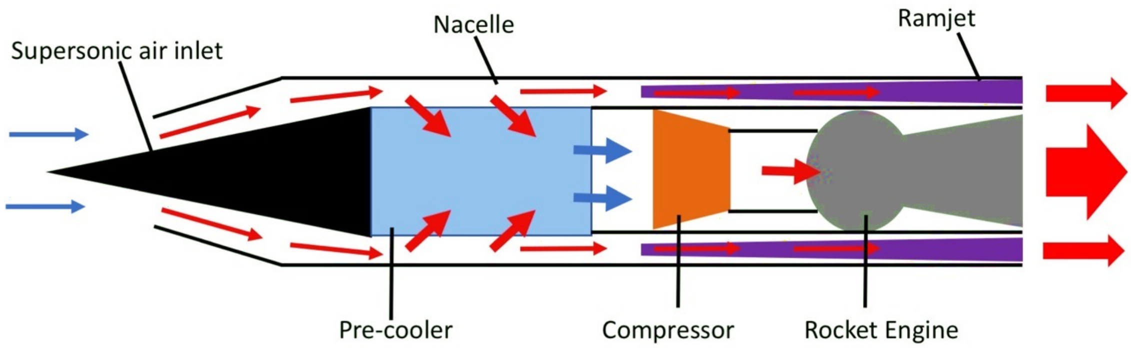


Figure 1.1 Skylon SABRE Engine Simplified Diagram

Seen in Figure 1.1 the SABRE (Synergistic Air-Breathing Rocket Engine) simplified diagram that demonstrates the air flow though the Engine such engine have a coaxial configuration, SABRE engine the coaxial engines configuration can be used with any types of Multiple engines (jet generators) integrated to achieve a flight that starts from subsonic to hypersonic going through supersonic speeds, such configuration is seen in Figure 1.2; in this figure the engine have a simple configuration of two engines or jet generators that share the same exit area and have a fixed circular engine cross-sectional area; this engine configuration is planned to be studied in the current research regardless of the type of engine been used such engine will be considered as a jet generator, however the exhausted flow out of these jet generators will always going to be considered as a supersonic jet, as in this research the only case that will be looked into is a supersonic jets exhausted into a hypersonic free stream flow, and the mixing between the two coaxial jets will be accomplished only when both jets are supersonic.

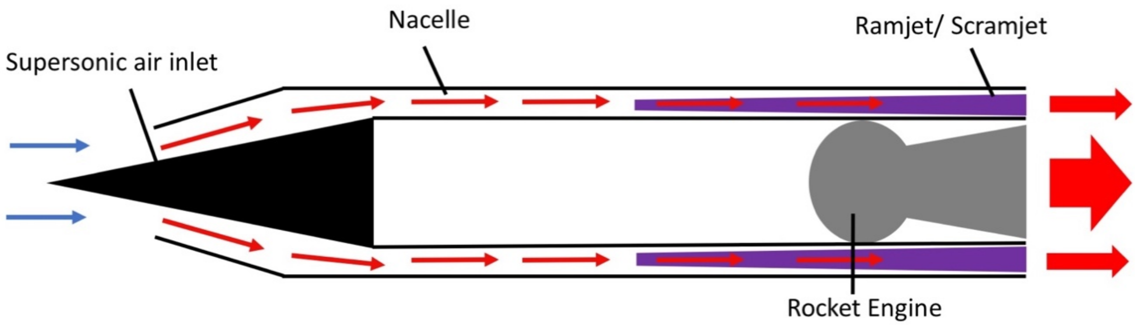
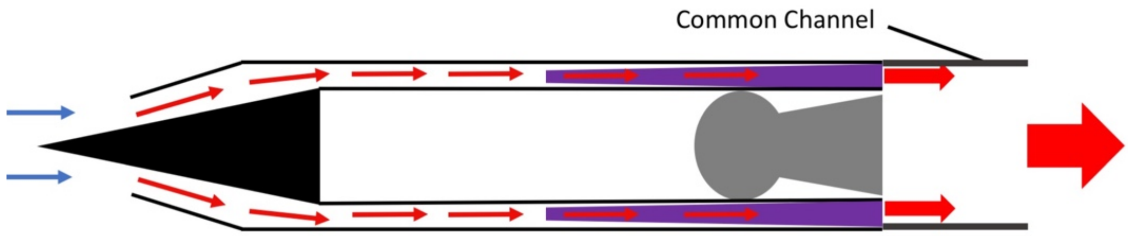


Figure 1.2 Multiple Engines Integration

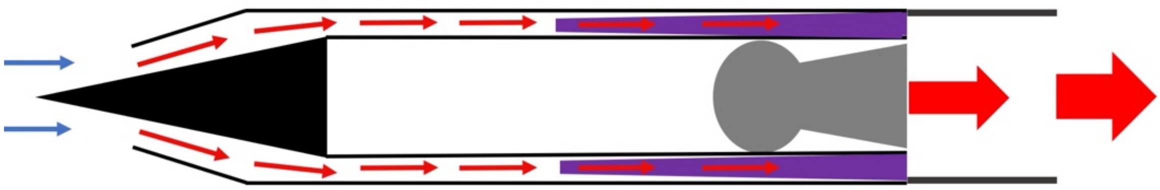
1.1.2 The Addition of Common Channel to a Coaxial Hypersonic Engine:

The main issue the current research is addressing; is the effect of adding a common channel to a coaxial supersonic jets generators configuration, when operating in single operational mode that is one of the jet generators are active, either the central or the surrounding only (case a and b in figure1.3), or both jets are in operation which can be refer to as a dual operational mode (case c in figure1.3). The main objective of current study is to evaluate the performance of co-axial nozzle in hypersonic environment and to study the effect of the common channel to enable thrust augmentation for various operation modes. The study is conducted by qualitative experimental flow visualization of shock structure for single and dual operation modes with common channel. Further, numerical simulations are performed for cases to evaluate the performance of system by thrust calculations. Further parametric study has been performed for resized model by isentropic as well as CFD simulations for various central and surrounding jet Mach numbers without common channel. Further, CFD simulations have been performed to study the effect of common channel on flow field and thrust performance.

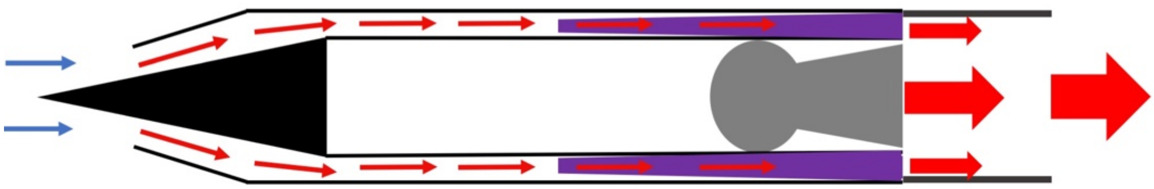
Coaxial engines exhaust jets flow has an impeded complexity in regards how to model the exhaust jet flows as well as the flow structure within and outside the body, not to mention the effect of using a cold airflow in the experiment to simulate the actual hot exhaust engines jet flow; In the current research the experimental model is an axisymmetric slender body having a jet generator in it, is consider simulating a combined cycle engine for the hypersonic flight. The jet generator is composed of two coaxial axisymmetric nozzles and a short common channel, in which two jets meet before being exhausted at the base of the body. Experiments have been performed for small slender body kept in hypersonic Mach 7 flow, which consists of two 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, called 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 were validated with experimental images. Later, the parametric study has been performed 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, that is Single central jet operating only, surrounding jet operation only, and dual operational mode in which both jets are in operation.



(a) Single operational mode: Surrounding jet only in operation



(b) Single operational mode: central jet only in operation



(c) Dual operational mode: both jets are in operation

Figure 1.3 Multiple Engines Integration Operational Modes With an added Common Channel

The common channel effect on performance and jet structure is studied in this research, within a hypersonic flight as the flow at this flight region is taking place a high altitude and the coaxial engines will produce an under-expanded jet; also, not to mention the effect of high speed hypersonic free stream shear layer on the coaxial engine jets. the main importance of studying the common channel in a hypersonic flow; that both the central and surrounding jets might be at supersonic exhausted flow speeds operational condition, in which both jets will be interacting in the common channel in the supersonic speed as well; for that mixing the coaxial jets in supersonic state for a flight hypersonic free stream speeds represents a challenge as the jet flows will generate shock waves as they get exhausted into the common channel, such case appears when the coaxial jets operating together and when one of the jets central or surrounding is only in operation, Figure 1.4 below shows the flow structure when both jets.

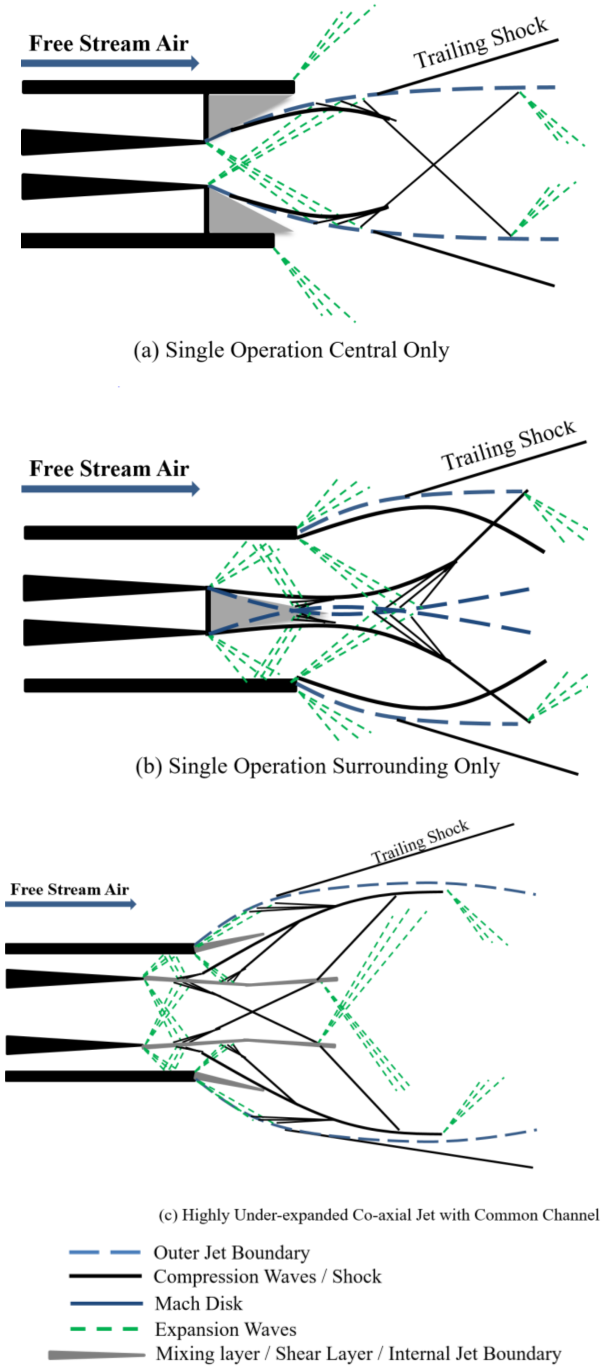


Figure 1.4 Schematic for (a) highly under-expanded single jet, (b) co-axial jet with common channel (c) co-axial jet without common channel

One of the issues that is expected when having a coaxial jet operating in a single operational condition that is central or surrounding jet only in operation is that the flow will have a recirculation flow in for the other jet cavity that will behave depending the kind of jet and the amount of expected flow blead from the other jet that is having a small amount of gas flow rate such as in the case of using a gas turbine; in this research for the purpose of simplifying the problem it is considered that when the single jet central or surrounding the other jet flow exit will be considered as a solid wall such condition will cause a flow circulating in the location of the jet that is not in operation such circulation can be seen in the figures 1.5.

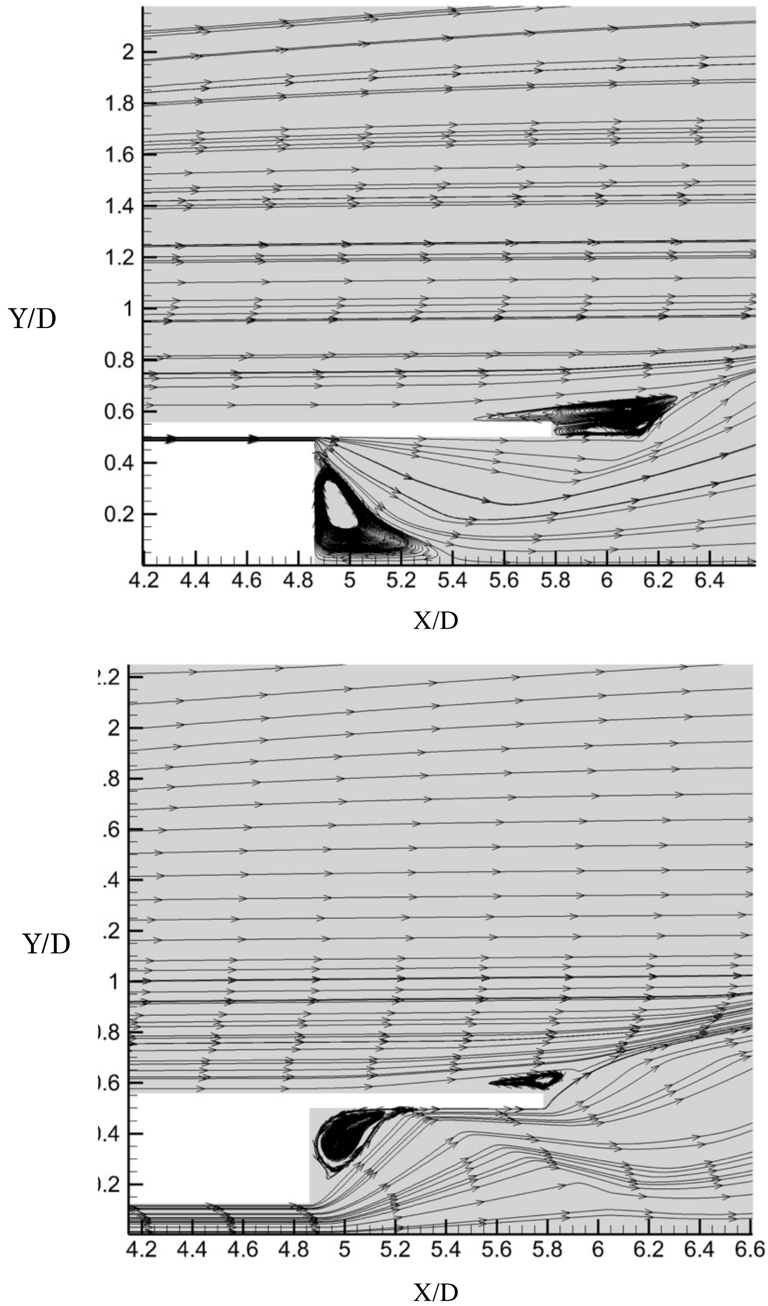


Figure 1.5: Flow circulation (a) M=1 Surrounding jet only in operation. (b) M=1 Central jet only in operation.

1.1.3 Experimental Methods to Simulate Engine Exhaust Jet in Wind Tunnel:

Experimental work have been performed to validate the CFD work that will be used to evaluate the common channel effect, such experimental work is not intended to be used to mimic an actual hypersonic flight. The main challenge of experimental modeling of the exhaust jet with integrated body of hypersonic vehicle prototype is the small size of the prototype, which has to be constructed within uniform core flow of test section. Hence, with the size of the prototype the size of exhaust nozzle has also be reduced, and in most of the facility it is not advisable to used hot flames in the test section. The modeled exhaust flow should be capable of generating a jet that can simulate real exhaust jet in hypersonic wind tunnel test section. The cold flow injection method has been chosen to model the jet flow in the hypersonic wind tunnel, such method will be the most convenient to be used as a result of it relative easiness to accomplish. Many ways that can be used to create a jet to be discharged out from a small aircraft model in a wind tunnel such as the following methods:

Mini-combustor Insertion: A mini-combustor faded with two fuel lines of oxygen and hydrogen, can be used along with the aircraft frame. Such idea will create a jet in the wind tunnel that consist of combustion products; but lack of nitrogen in the combustion products will come as a result of pumping both the fuel and the oxidizer and not using the air in the wind tunnel to achieve combustion in such small scale, on the other hand it is highly expensive and hard to fabricate and design, along of being unpractical if used in small test section hypersonic wind tunnels.

Hot Combustion products injection: Instead of performing combustion in a small hypersonic vehicle model; the combustion can be performed separately and its products can be collected, that the exhaust flow will mainly consists of (H_2O, N_2, NO, Ar, H_2); to be used as a gas that will be heated and then injected in the wind tunnel, the main disadvantage of such a method is the required facility capable of performing and collecting the combustion products.

Hot H_2O, N_2 gas injection: In this method, we inject a mixture of water and nitrogen as a hot stream of gas to the wind tunnel model, such method is highly accurate when compared to the hot combustion products injection.

Cold Mixed (Ar and CF_4) flow injection: One of the highly practical and accurate ways to model a jet flow in a small test section hypersonic wind tunnel, is to inject a mixture of cold gases such as Ar and CF_4; that the value of the normalized thrust, normalized lift, and normalized pitching moment, is within high accuracy to the hot combustion products injection. The main problem of using such method is the use of CF_4 that can be toxic to biological entities and harmful for environment, which limits the use of such method.

Hot air injection: to inject air flow at high temperature (total temperature of 5200K) and high pressure (Nozzle pressure ratio [NPR] of 3000) [21], to the model to be discharged to the hypersonic outer flow conditions, this method is relatively simple. However, it has an extra complexity over the cold flow method, as we need to heat the flow up; which need a device or a mechanism to perform it.

Cold air injection: Air is injected at relatively high pressure (NPR=3000), but relatively low temperature [21], to the model to expand in the hypersonic free stream conditions, this method is highly practical and easy to use in many deferent pressure settings, however it has the poorest accuracy within the methods discussed previously.

Based on Kenneth, Tatum, Lawrence, and Huebner paper "Exhaust Gas Modeling Effects on Hypersonic Powered Simulation at Mach 10" [21] through comparing the methods been discussed and evaluating the advantages and the disadvantages of using one method rather than the other, table 1.1 can be constructed. The cold flow injection method has been chosen to model the jet flow in the hypersonic wind tunnel, such method will be the most convenient to be used as a result of it relative easiness to accomplish. As have been seen in the previous discussion, that there are many ways to simulate a jet with deferent accuracy levels, the choice of witch one to use will always depend on the stage of the study and the needed accuracy; in this research we are studying the general behavior and what affect its general performance parameter the use of the cold flow will provide us the needed accuracy as we are not planning to build an experimental prototype that will be tested in actual flow field such as launching a hypersonic aircraft experimental project.

Jet Modeling method	Advantages	Disadvantages
Mini-combustor Insertion.	- Actual combustion occurrence.	- lack of nitrogen. - highly expensive. - hard to fabricate.
Hot Combustion products injection.	- High accuracy in comparison with the actual combustion case	-Facility capable of performing and collecting the combustion products. - Gas heating is needed.
Hot H2O, N2 gases injection.	- high accuracy compared to the combustion products injection. - Limited species injection.	- Gas heating is needed.
Cold Mixed flow (Ar and CF4) injection.	- Highly practical and accurate. - No heating is needed.	- Toxic CF4 is needed.
Hot air injection.	- Air usage.	- Gas heating is needed. - Relatively low accuracy.
Cold air injection	- Easy to perform. - Low cost.	- Poor accuracy.

Table1.1 Hypersonic wind tunnel jet modeling methods

1.2 Literature Survey

Co-axial engine configuration have been applied in the SR71 black bird engines [11][12], which integrates the Pratt and Whitney J58 engine with a ramjet engine, such configuration also holds a key features discussed in the paper of Peter W. Merlin with the title of " Design and Development of the Blackbird: Challenges and Lessons Learned". One of the study of co-flowing supersonic over expanded jets with central Mach number 1.96 have also been performed in open-jet facility [13].and found that jet core length increases in presence of co-flow.In this study, with increase of surrounding jet speed up to under-expanded co-flow operation with supersonic central-jet, complicated interactions among the shock-cells have also been observed. However, conducting such studies in wind tunnel environment along with front body is rare. A recent study on the Skylon Airplane and its SABRE engines Plumes [14] simulates 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.

A part of the effect of surrounding flow-field on the exhaust jet, the performance of co-axial supersonic jet exhaust depends on common channel, which can be theoretically considered as sudden expansion region because of available area increase for flow after exiting from nozzles in single and dual operation modes. Supersonic flow exiting in suddenly expanded region have been studied by Bulat et al. [15] and classified different flow regime as steady, oscillating and transient, based on interaction of wall and re-circulation regions. Further, the effects of suction and blowing on flow Separation in a symmetric suddenly expanded channel is also examined by Layek, et al. numerically [16]. The suddenly expanded nozzle is well studied for the case of single jet expanding in higher area channel. However, the effect of surrounding jet on a suddenly expanded central jet in hypersonic environment has not been studied in the literature according to the knowledge of authors. The current study will evaluate the performance of common channel in single operation mode only central jet expanding in common channel and the effect of surrounding jet in dual operation mode when surrounding jet may be used as controlling the sudden expansion by blowing.

Axisymmetric underexpanded jet has been mainly studied for free jets by discharging it into still atmospheric conditions [6] such studies examine the basic features of under expanded, correctly expanded, and over expended jets flow; such studies emphasize on the jet flow field which includes the jet shape and its shock wave structures. There are also some studies that deals with underexpanded axisymmetric jets in the hypersonic regime numerically [7] by solving implicit Navier-Stokes equations applied to hypersonic exhaust plume/afterbody flow fields. There is also some studies that simulate a single undersxpanded jet [8] such study utilizes the PIV technique to visualize the flow field of the jet flow as well as its interaction with the free stream flow and the wake flow behind a body facing a Mach 3 flow condition.The study of Supersonic jets flows and it interactions can be seen throughout the literature such as studying a twin or more interaction of jets [9] entitled of "CFD Analysis of Twin Jet Flow At Mach 1.74.with Fluent Software". The Nozzle performance of an exhaust that has internal and external flow expansion has been studied too [10] this study uses ANSYS Fluent to perform the CFD and validate it by comparing it with experimental data, such study focuses on the shock waves reflection and its effect on the pressure distribution on the ramp of the after-body.

Historically supersonic under-expanded jets exhausted into a still air have been studied throughout the literature, one of the oldest yet interesting studies have been conducted is performed by S.Crist, P.Sherman, and D.Glass in the year of 1965 [22]; such study discuss the highly underexpanded sonic jet, in this study the effect of a high pressure ratio on the properties of the Mach disk being formed along with the Mach number distribution across the jet axis, an interesting finding in this study is that at high pressure ratios the ratio between the Mach disk diameter and the Mach disk distance appeared to be constant for a given gas. Also in 1974 [23] .Chang, and W.Chow, studied the flow structure of the underexpanded free jet flow and the formation of the Mach disk along with its size and location. The Length of the Supersonic Core in High Speed Jets have been investigated in Shirie, and Seubold in 1967 [24], The noise of the jet was always a concern and can be found in the Journal of Sound and Vibration in a paper published in 1982 [25] by C.Tam, and H.Tanna, such studies the noise associated with the supersonic jets from a convergent-divergent nozzle. One study that combines Experimental work along with a solution of the parabolized Navier Stokes equation to study the structure of turbulent sonic underexpanded free jets is done by S. Chuech, M.Lai, and G. Faeth in May, 1989 [26], such study performers test of free under-expanded sonic jet with an under-expansion ratios of less than 1.4.

Modeling a jet plumes in a wind tunnel is examined in 1980 [27] by Korst, White, Nyberg, and Agrell in which they showed there work with a paper entitled ‘The simulation and modeling of jet plumes in wind tunnel facilities’, such paper introduces to an experiment made in a supersonic wind tunnel by creating a high pressure hot gas supply to create a jet inside the wind tunnel, to study the aerodynamic interface effects caused by plume induced flow separation from a propulsive after body, the main aim of the experiment is to evaluate the merits and potential of a proposed plume simulation methodology that is based on second order geometrical modeling schemes, such work is followed by a paper by also H.Korst, and R.White in 1981 with the same research interest [29] ; a detailed description about similar experimental work is descriped by Nyberg, Agrell, and Heverng in thiere 2nd Annual Technical Report with the title of ‘Investigation of Modeling Concepts For Plume-Afterbody Flow Interactions’ issued in 1980 [28]. G. Deiwert performed A Computational Investigation of Supersonic Axisymetric Flow Over Boattails Containing a Centered Propulsive Jet on the year of 1983, which is a CFD study based on solving the Reynolds-Avereged Navier Stokes Equations for compressible flow, such study conceders a Mach 2 free stream flow with a jet exiting with Mach number of 2.5. [30, 31]. It can be also seen some studies related to the External flow Aerodynamics of a missile interaction with a jet plume at it aft in 1987, 1992, and 1999 in the following references [32], [33]. An Interesting research that deals with aerodynamics of misslile afterbody with ring nozzle in trans and supersonic stream, in this research an experiment of a ring under-expanded and over-expanded jet with external flow stream is being investigated regarding to the flow structure that is being produced, such research took place in 1999, in the Moscow State Aviation Institute. [34] Coaxial jets exhausting in a still air have been studied such as in Pinnam, and Rathakrishnan, [35] in which it presents an experimental studies on the effect of annular Co-Flowing Subsonic and sonic jets, for jets with under expanded and correctly expanded conditions the summery of this research is that by adding an annular jet to the central jet the central jet gets more propagated in which without such annular jet the central jet will decay in a smaller axial distance. A sophisticated test have been performed to install a Hydrogen Fueled detonation ramjet engine model into a pulse wind tunnel for a free stream conditions of Mach 4, 5m 6, and 8, such study is mainly focused on measuring the thrust and Isp performance of the ramjet engine [36]. Sudden expansion jet expanding into a channel have been investigated by Ashfaq, and Khan, in such paper [37] discuss the results acquired through experiments for a convergent nozzle for subsonic sonic and sonic under expanded flow.

In recent studies, a coaxial jets in hypersonic flow related studies can be viewed such as Murli et al. [38] (2018) who have studied the interaction of twin sonic jets in parallel and inward canting configuration. Safir et al. in Belkacem, and Beghifja [39] (2019) Numerical Investigation of Coaxial Turbulent Jet have been conducted from the view point of investigating laminar to turbulent transition of coaxial subsonic jets discharging into static air. Stephan, and Radespiel at DLR [40] (2012), 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. A NASA Technical Report with the title of “A Supersonic Argon/Air Coaxial Jet Experiment for Computational Fluid Dynamics Code Validation” [41] (2007) 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 performed by Baurle and Edwards [42] (2009) have studied mixing of Helium-air and argon-air for same co-axial nozzles of Clifton et al [43] (2019) exiting in open atmosphere by using hybrid RANS/LES simulations. Zaman et al. The paper of the title of “Wind Tunnel Model Design for Sonic Boom Studies of Nozzle Jet with Shock Interaction”[44] (2016) have conducted experimental and computational study of various tones occurring in supersonic designed co-axial annular nozzles operating in range of low Mach numbers. Recently, Ivanov et al. [45] (2018) experimentally investigates 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.

Suddenly expanded nozzle is well studied for the case of single jet expanding in higher area channel. However, the effect of surrounding jet on a suddenly expanded central jet in hypersonic environment has not been studied in the literature according to the knowledge of authors. The current study will evaluate the performance of common channel in following cases: single operation mode with central jet expanding in the common channel, surrounding jet expanding in the common channel, and the effect of the common channel when both jets injected in the common channel. The main objective of current study is to evaluate the performance of co-axial nozzle in hypersonic environment and study the effect of the common channel to enable thrust augmentation for various operation modes. The study is conducted by qualitative experimental flow visualization of shock structure for single and dual operation modes with common channel. Further, numerical simulations are performed for cases to evaluate the performance of system by thrust calculations. Further parametric study has been performed for resized model by isentropic as well as CFD simulations for various central and surrounding jet Mach numbers without common channel. Further, CFD simulations have been performed to study the effect of common channel on flow field and thrust performance.

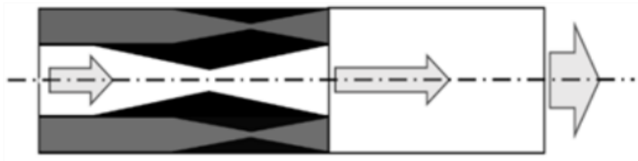
Single and Coaxial engines jet interaction has been sufficiently studied in both the subsonic and the supersonic flow regimes; we cannot find as much studies at the hypersonic regime, although such topic has critical importance; taking into consideration that the exhaust flow of the hypersonic aircraft experience a significant expansion at the aft-body, and when we have two or more jets expanding at the aft part of the aircraft or the region downstream of the engine; such interaction between them is going to strongly affect the performance of the aircraft.

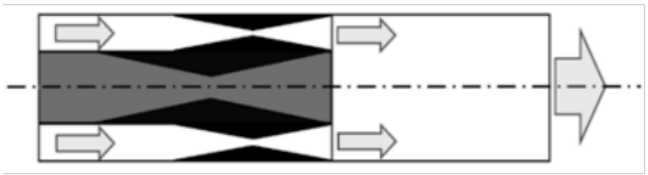
As can be seen throughout the literature the hypersonic aircraft and its engines integration is a new research topic, and it is not seen in the literature a sufficient studies that deals with single and coaxial engine exhausts jets interaction discharged from a hypersonic body such study is extremely important as it provides a practical method to achieve a powered hypersonic flight.

1.3 Objectives

The current study focuses on 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. Also, in order to study the augmentation of total thrust from both the nozzles, the wall of outer nozzle have been extended so that central and surrounding jets interact in a confined passage before exhausted into the low-pressure hypersonic external flow. In this study, this confined passage is termed as common channel.

Figure 1.6 depicts the single and 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 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 length of straight passage can also add the weight and can increase the length of vehicles.


(a) Single Operation Mode (Central-Jet)


(b) Single Operation Mode (Surrounding-Jet)

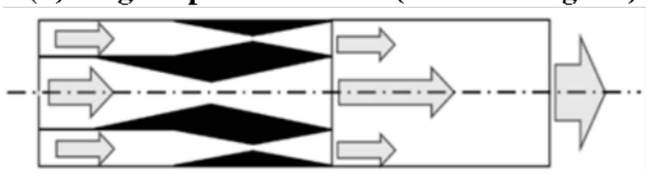

(c) Dual Operation Mode (Both-Jets)

Figure1.6 Coaxial configuration operational modes with common channel.

It is also important to understand the effect of length of such common channel on thrust for Single (Central only or Surrounding only) and dual operational modes to optimize the length; as the longer extended common channel can also lead to reflection of expansion waves from the wall as compression front and depends on expansion level, Mach disk may also form before the end of common channel, which can reduce the performance. Hence, apart from weight increase, the flow-field inside the common channel may also have impact on performance of whole system in dual and single operation modes. Fundamentally, optimum length of common channel can improve thrust performance by reducing the expansion level in each single operation mode and provide some area to augment thrust in dual operation mode; the following are the research main objectives and its originality points from past research studies.

Research Main Objectives

- Numerically model the flow of coaxial supersonic jets, with a comparison with Hypersonic experiments.
- Understand the flow field of the coaxial supersonic jets in hypersonic free stream flow, and the effect of adding a common channel to such coaxial supersonic flow.

- Study the effect of common channel on performance, for both single and dual modes of operation.
- Investigating the most effective area division between the central and surrounding engine exhaust exit areas when both jets generated are at an under-expanded state.

Current study main originality points:

1- Study the suddenly expanded flow through a channel, by expanding the flow from a surrounding ring nozzle towards the central axis of the nozzle, and compare such case with the traditionally studied case of expanding the flow from a central circular nozzle towards the channel.

2- Understand the effect of the common channel on flow field and performance when operating in a dual mode with both central and surrounding jets active.

3- Understand the effect on performance of adding a common channel to a coaxial jet configuration with relation to its modes of operation.

The overall objective of current study is to evaluate the performance of coaxial nozzle in hypersonic environment and the effect of the common channel to enable thrust augmentation for various operation modes. The study is conducted by qualitative experimental flow visualization of shock structure for single and dual operation modes with common channel. Further, numerical simulations are performed for cases to evaluate the performance of system by thrust calculations. Further parametric study has been performed for resized model by isentropic as well as CFD simulations for various central and surrounding jet Mach numbers without common channel. CFD simulations have been performed to study the effect of common channel on flow field and thrust performance.

1.4 Thesis Outline

The content of the current thesis will start by the first chapter introducing the common channel and its application to the coaxial engine configuration in order to augment the trust without an increase in both jets cross sectional area. Chapter 2 is presenting the Compressible flow simulation method to solve the Navier-Stokes equations in the axisymmetric configuration, the solver specifications will be discussed which is a generalized 2-D axisymmetric equations solver that solves the inviscid part of the Compressible axisymmetric Navier Stokes equations by using Liou's all speed AUSM +up scheme [10], with a 3rd-ordered upwind biased MUSCL interpolation along with an Entropy fixation option. The 2nd ordered central difference method have been used for solving the viscous terms. The solver also utilizes the 3 step TVD Runge Kutta for time marching. Chapter 3 which presents the experimental method such method has been useful in performing studies on both the single jet as well as the coaxial jet configuration, in this section the experimental set up will be explained from how to prepare the experiment, design the models, to how to manufacture and assemble them, this section also contains the experimental procedure and the method of collecting the data. Chapter4 discusses a comparison between experimental and numerical Results Qualitatively to evaluation of the flow field, and Quantitatively evaluate the pressure at the common channel as it represents the main point of interest. Both chater5 and chapter 6 show the main results of the current research regarding the effect of common channel on flow fiel and the effect of common channel on performance coaxial jets performance. The aim of this Chapter 5 is to identify the fundamental characteristics of the interactions between the two streams of the coaxial jets; and to assess the interaction between the central and surrounding jet flows and the free stream flow running around the body flying at hypersonic speed. In this chapter 6 a 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 operating exit Mach numbers on performance in single and dual operation modes have been studied with the thrust evaluation and flow-field analysis. In the last and third part, the effect of addition of various common channel lengths on the flow-field and the thrust performance of the system have been evaluated by numerical simulations. Finally at chapter 7 some concluding remarks are been summarized and proposed future works are mentioned, Below is a summary of the thesis and the content of each chapter:

Chapter 1: Study Objectives, Motivation, and Previous research

Motivation:

To study the effect of adding a common channel to a coaxial jet in order to augment the trust without an increase in both jets cross sectional area.

Objectives:

(1) Understand under-expanded jets flow into a hypersonic free stream flow.

(2) Investigate the effectiveness of adding a common channel to a coaxial jets operating in different conditions as central only surrounding only or both jets in operation.

Previous Research:

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. The current 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.

Chapter 2: Numerical Methods

Governing Equations:

Discussing The governing equations been solved in the numerical simulation CFD model; which is Compressible axisymmetric Navier Stokes equations, flow is considered to be a laminar flow.

Numerical Model:

Discussing the Numerical model been used to solve the govering equations in which the solver is a generalized 2-D axisymmetric equations solver that solves the inviscid part of the Compressible axisymmetric Navier Stokes equations by using Liou's all speed AUSM+upscheme, with a 3rd-ordered upwind biased MUSCL interpolation along with an Entropy fixation option. The 2nd ordered central difference method have been used for solving the viscous terms. The solver also utilizes the 3 step TVD RungeKutta for time marching. the main reason of choosing the AUSM+up scheme.

Chapter 3: Experimental method:

The experiments were conducted in Kashiwa Hypersonic Wind Tunnel Facility at Graduate School of Frontier Sciences, The University of Tokyo with a model that have supersonic coaxial jets generator.

Purpose:

The Experimental work main objective is to provide the data needed to compare the flow field generated for experiment and the one generated by the CFD results. Two sets of experiments are mainly used, the first is to high resolution images that Capture a static picture and synchronizing it with a full body flow field and a jet focused image, along with record the jets chambers pressures and a pressure reading measured at the common channel.

Chapter 4: Comparison of Experimental and Numerical Results

Experiments have been used to be compared with the CFD results by the following parameters:

1- Qualitatively evaluation of the flow field.

2- Quantitatively evaluate the pressure at the common channel as it represents the main point of interest.

Chater5: Coaxial Exhaust Jets Flow Field Analysis

The aim of this Chapter is to identify the fundamental characteristics of the interactions between the two streams of the coaxial jets; and to assess the interaction between the central and surrounding jet flows and the free stream flow running around the body flying at hypersonic speed. The flow is featured by having two jets, two shear layers, a central Oblique shock wave, and an external trailing shock wave as a result of the interaction between the coaxial jet and the free stream flow around the hypersonic body.

In this section, initially flow features of co-axial supersonic jets in hypersonic environment are discussed for single and dual mode operations for experimental models and corresponding CFD simulations. In the later part, the study of the effect of the common channel on flow field is evaluated by using CFD simulations for the study model that been discussed before.

Chapter 6: Common Channel Performance Analysis

In this chapter a 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 operating exit Mach numbers on performance in single and dual operation modes have been studied with the thrust evaluation and flow-field analysis. In the last and third part, the effect of addition of various common channel lengths on the flow-field and the thrust performance of the system have been evaluated by numerical simulations.

Chapter 7: Conclusion and Future Works

is found that the introduction of 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 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.

For the future work it is recommended to study the process of starting the jet flow as in starting the central jet after that starting the surrounding jet, such starting process can have an interesting behavior of jets as it get through from starting phase to steady state phase. For the common channel it is also interesting to study the effect of the common channel as it operates through different free stream Mach numbers from subsonic, supersonic to hypersonic flow.

Chapter 2

Numerical Methods

The governing equations been solved in the numerical simulation CFD model is the Compressible axisymmetric Navier Stokes equations, the flow is considered to be a laminar flow because the jet flow is supersonic thus the shock structure will have the main impact on the flow field and not the fluid turbulence. the solver it is a generalized 2-D axisymmetric equations solver that solves the inviscid part of the Compressible axisymmetric Navier Stokes equations by using Liou's all speed AUSM+upscheme[17], with a 3rd-ordered upwind biased MUSCL interpolation[18] along with an Entropy fixation option. The 2nd ordered central difference method have been used for solving the viscous terms. The solver also utilizes the 3 step TVD RungeKutta[19] for time marching. the main reason of choosing the AUSM+up scheme is that in the present computation we have a flow that will have a regions which is supersonic and subsonic in which and in the case of shock wave passing through the subsonic region an unphysical numerical viscosity may appear; thus this numerical scheme is advised to be used in such computations.

2.1 Axisymmetric Navier StokesEquations

Conservative form of the Compressible axisymmetric Navier Stokes equations is shown below, such equations comprises of three parts, inviscid, viscous, and source therm.

$$\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + H = \frac{\partial E_v}{\partial x} + \frac{\partial F_v}{\partial y} + H_v \tag{2.1}$$

Where:

$$Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ E_t \end{bmatrix} E_t = \rho e + \frac{1}{2}\rho(u^2 + v^2) \tag{2.2}$$

$$E = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ (E_t + p)u \end{bmatrix} \quad , \quad F = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (E_t + p)v \end{bmatrix} \quad , \quad H = \frac{1}{y} . \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 \\ (E_t + p)v \end{bmatrix} \quad , \quad E_v = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ u\tau_{xx} + v\tau_{xy} + \dot{q}_x \end{bmatrix} \tag{2.3}$$

$$F_v = \begin{bmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ u\tau_{xy} + v\tau_{yy} + \dot{q}_y \end{bmatrix} \quad , \quad H_v = \frac{1}{y} \begin{bmatrix} 0 \\ \tau_{xy} \\ 2\mu\left(\frac{\partial v}{\partial y} - \frac{v}{y}\right) \\ u\tau_{xy} + v\tau_{yy} + \dot{q}_y \end{bmatrix} \tag{2.4}$$

$$\tau_{xx} = \frac{2}{3}\mu\left(2\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{v}{y}\right) \tag{2.5}$$

$$\tau_{yy} = \frac{2}{3}\mu\left(2\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} - \frac{v}{y}\right) \tag{2.6}$$

$$\tau_{xy} = \mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \tag{2.7}$$

$$\dot{q}_x = \kappa\frac{\partial T}{\partial x} \quad , \quad \dot{q}_y = \kappa\frac{\partial T}{\partial y} \tag{2.8}$$

The flow kinematic viscosity will vary according temperature according to.

$$\mu = \frac{c_1(T^{1/2})}{1 + c_2 / T}, \tag{2.9}$$

$$c_1 = 1.458 \times 10^{-6} Kg / (m.s.K^{1/2}) \tag{2.10}$$

$$c_2 = 110.4K \tag{2.11}$$

2.2 Generalized Governing Equations

The governing equations solved in the computation are transformed into the dimensionless form, in accordance to the reference values listed below.

$$
\begin{aligned}
r_{ref} &= 1.0 \\
v_{ref} &= u_\infty \\
\rho_{ref} &= \rho_\infty \\
T_{ref} &= T_\infty \\
\mu_{ref} &= \mu_\infty \\
p_{ref} &= \rho_\infty u_\infty^2 \\
q_{ref} &= \rho_\infty u_\infty^3
\end{aligned}
\tag{2.12}
$$

Non dimensionalized parameters are defined in accordance to the reference values.

$$
\begin{aligned}
x^* &= \frac{x}{r_{ref}} \\
y^* &= \frac{y}{r_{ref}} \\
u^* &= \frac{u}{v_{ref}} \\
v^* &= \frac{v}{v_{ref}} \\
\rho^* &= \frac{\rho}{\rho_{ref}} \\
T^* &= \frac{T}{T_{ref}} \\
\mu^* &= \frac{\mu}{\mu_{ref}}
\end{aligned}
\tag{2.13}
$$

The equations has been dimensionalized and generalized to be solved by using a finite difference method, the dimensionalized and generalized system of equations are.

$$\frac{\partial \hat{Q}}{\partial t}+\frac{\partial(\hat{E}-\hat{E}_v)}{\partial \xi}+\frac{\partial(\hat{F}-\hat{F}_v)}{\partial \eta}+\frac{(\hat{H}-\hat{H}_v)}{y}=0 \tag{2.14}$$

Where:

$$\hat{Q}=\frac{1}{J}Q \tag{2.15}$$

$$\hat{E}=\frac{\xi_x}{J}E+\frac{\xi_y}{J}F \tag{2.16}$$

$$\hat{F}=\frac{\eta_x}{J}E+\frac{\eta_y}{J}F \tag{2.17}$$

$$\hat{H}=\frac{1}{J}H \tag{2.18}$$

$$\hat{E}_v=\frac{\xi_x}{J}E_v+\frac{\xi_y}{J}F_v \tag{2.19}$$

$$\hat{F}_v=\frac{\eta_x}{J}E_v+\frac{\eta_y}{J}F_v \tag{2.20}$$

$$\hat{H}_v=\frac{1}{J}H_v \tag{2.21}$$

$$\frac{1}{J}=\begin{vmatrix} x_\xi & y_\xi \\ x_\eta & y_\eta \end{vmatrix},\quad \begin{bmatrix} \xi_x & \eta_x \\ \xi_y & \eta_y \end{bmatrix}=J.\begin{bmatrix} y_\eta & -y_\xi \\ -x_\eta & x_\xi \end{bmatrix} \tag{2.22}$$

$$Q=\begin{bmatrix} \rho \\ \rho u \\ \rho v \\ E_t \end{bmatrix},\quad E=\begin{bmatrix} \rho u \\ \rho u^2+p \\ \rho uv \\ (E_t+p)u \end{bmatrix},\quad F=\begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2+p \\ (E_t+p)v \end{bmatrix},\quad H=\begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 \\ (E_t+p)v \end{bmatrix} \tag{2.23}$$

$$E_v = \frac{1}{\mathrm{Re}} \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \beta_x \end{bmatrix}, \quad F_v = \frac{1}{\mathrm{Re}} \begin{bmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ \beta_y \end{bmatrix}, \quad H_v = \frac{1}{\mathrm{Re}} \begin{bmatrix} 0 \\ \tau_{yx} \\ \alpha_y \\ \beta_y \end{bmatrix} \tag{2.24}$$

$$\tau_{xx} = \frac{2}{3}\mu\left(2\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{v}{y}\right) \tag{2.25}$$

$$\tau_{yy} = \frac{2}{3}\mu\left(2\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} - \frac{v}{y}\right) \tag{2.26}$$

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \tag{2.27}$$

$$\alpha_y = 2\mu\left(\frac{\partial v}{\partial y} - \frac{v}{y}\right) \tag{2.28}$$

$$\beta_x = \tau_{xx}u + \tau_{xy}v + \frac{\mu}{\Pr M^2(\gamma-1)}\frac{\partial T}{\partial x} \tag{2.29}$$

$$\beta_y = \tau_{yx}u + \tau_{yy}v + \frac{\mu}{\Pr M^2(\gamma-1)}\frac{\partial T}{\partial y} \tag{2.30}$$

$$M = \frac{V_\infty}{\sqrt{(\gamma-1)C_p T_\infty}}, \quad \mathrm{Re} = \frac{\rho_\infty L V_\infty}{\mu_\infty}, \quad \Pr = \frac{C_p \mu_\infty}{k_\infty} \tag{2.31}$$

2.3 General Discretization

The equations are discretized in accordance with the following.

$$Q_{i,j}^{n+1} = Q_{i,j}^{n} + (\lambda R(Q_{i,j}^{n+1}) + (1-\lambda)R(Q_{i,j}^{n})) \tag{2.32}$$

$$\begin{aligned} R(Q_{i,j}^n) = -\Delta t(&E_{i+1/2,j}^n - E_{i-1/2,j}^n + F_{i,j+1/2}^n - F_{i,j-1/2}^n + Ev_{i+1/2,j}^n \\ &+ Ev_{i-1/2,j}^n + Fv_{i,j+1/2}^n + Fv_{i,j-1/2}^n + H_{i,j}^n - Hv_{i,j}^n) \end{aligned} \tag{2.33}$$

The finite difference of the variable Q is evaluated by the above mentioned equations in which the value of λ can change to acquire an explicit or implicit scheme, in the current computation the value of λ is set to 0 that will make the scheme an explicit one. E,and F are the numerical fluxes that is been calculated using usingLiou's all speed AUSM+up scheme, with a 3rd-ordered upwind biased MUSCL interpolation, the flux of Ev, and Fv they are been evaluated by the central difference method. Finally the source terms H, and Hv have been calculated directly.

2.4 Numerical Fluxes

MUSCL (Monotone Upstream-centered Schemes for conservation Law) has been used to evaluate the physical parameters at both sides of computed cell with taking the values of κ to be 1/3, to achieve an upwind-biased third-order scheme and the value of ε of 10^-16, to prevent a division of 0 for the values of ($Q_{i+1}-Q_i$) and (Q_i-Q_{i-1}).

$$Q^{+}_{i+1/2}=Q_i+\left\{\frac{s}{4}\left[(1-\kappa s)\Delta_- +(1+\kappa s)\Delta_+\right]\right\}_i \tag{2.34}$$

$$Q^{-}_{i+1/2}=Q_i+\left\{\frac{s}{4}\left[(1-\kappa s)\Delta_+ +(1+\kappa s)\Delta_-\right]\right\}_{i+1} \tag{2.35}$$

$$s=\frac{2\Delta_+\Delta_- +\varepsilon}{(\Delta_+)^2+(\Delta_-)^2+\varepsilon} \tag{2.36}$$

$$(\Delta_+)_i=Q_{i+1}-Q_i$$

$$(\Delta_-)_i=Q_i-Q_{i-1}$$

Liou's all speed AUSM+up scheme is used to evaluate the inviscid flow part of the non dimensionalized generalized Compressible axisymmetric Navier Stokes equations, such scheme will split this part of the Navier Stokes governing equations into a properties term and a pressure term demonstrated at the following equation 2.8.

$$E^n_{i+1/2}=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ (E_t+p)u\end{bmatrix}=\begin{bmatrix}\rho u\\ \rho u\\ \rho uv\\ e\end{bmatrix}u_{i+1/2}+\begin{bmatrix}0\\ p_{i+1/2}\\ 0\\ 0\end{bmatrix}=Ec_{i+1/2}+P_{i+1/2} \tag{2.37}$$

The properties term can be rewritten in terms of the Mach number that will be split into two portions lift and right such splitting can be demonstrated by the following equations:

$$Ec_{i+1/2} = \begin{bmatrix} \rho u \\ \rho u \\ \rho uv \\ e \end{bmatrix}_{i+1/2} \quad u_{i+1/2} = \begin{bmatrix} \rho c \\ \rho cu \\ \rho cv \\ \rho ch \end{bmatrix}_{i+1/2} \quad M_{i+1/2} \tag{2.38}$$

$$= M_{i+1/2} \begin{bmatrix} \rho c \\ \rho cu \\ \rho cv \\ \rho ch \end{bmatrix}_L \quad if, M_{i+1/2} \geq 0 \tag{2.39}$$

$$= M_{i+1/2} \begin{bmatrix} \rho c \\ \rho cu \\ \rho cv \\ \rho ch \end{bmatrix}_R \quad if, M_{i+1/2} \leq 0 \tag{2.40}$$

Mass flux and the properties of the lift and right side of the computed cell has to inserted To compute the flux of E using the below equation

$$E_{i+1/2} = \frac{\dot{m} + |\dot{m}|}{2}\psi^{+} + \frac{\dot{m} + |\dot{m}|}{2}\psi^{-} + \tilde{P}N \tag{2.41}$$

Where:

$$\psi = \begin{bmatrix} 1 \\ u \\ v \\ h \end{bmatrix}, \quad N = \begin{bmatrix} 0 \\ n_x \\ n_y \\ 0 \end{bmatrix} \tag{2.42}$$

Applying the AUSM+Up scheme requires a system of equations, Figure 2.1 shows the process in which the computation is been proceeded.

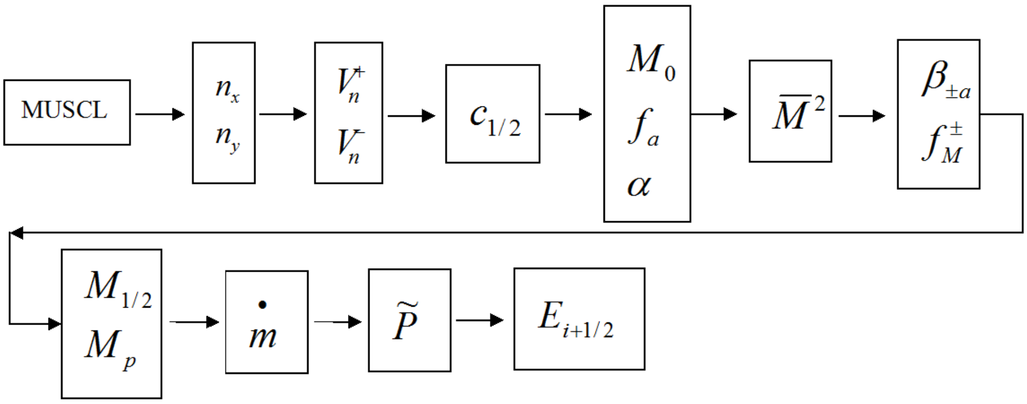


Figure 2.1 Governing equation inviscid part computation process

By using the governing equation inviscid part computation process the inviscid flow part the Navier Stokes governing equations is evaluated by applying the following set of equations listed below.

$$\widetilde{P} = \beta_{+a} P_L + \beta_{-a} P_R + P_u \tag{2.43}$$

$$\beta_{\pm a} = \left\{ \begin{array}{l} \frac{1}{2}(1 \pm sign(M)), \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots if |M| \geq 1 \\ \frac{1}{4}(M \pm 1)^2 (2 \mp M) \pm \alpha M (M^2 - 1)^2, otherwise \end{array} \right\} \tag{2.44}$$

$$P_u = -K_u \beta_+ \beta_- (\rho_L + \rho_R)(f_a c_{1/2})(V_n^- - V_n^+) \tag{2.45}$$

$$\begin{aligned} c_{1/2} &= \min(\widetilde{c}_L, \widetilde{c}_R), \\ \widetilde{c}_L &= c^{*2} / \max(c^*, V_n^+), \\ \widetilde{c}_R &= c^{*2} / \max(c^*, V_n^-), \end{aligned} \tag{2.46}$$

$$c^{*2} = \frac{2(\gamma - 1)}{(\gamma - 1)} h \tag{2.47}$$

$$\alpha = \frac{3}{16}(-4 + 5 f_a^2) \tag{2.48}$$

$$f_a(M_0) = M_0 (2 - M_0) \tag{2.49}$$

$$M_0^2 = \min(1, \max(\overline{M}^2, M_\infty^2)) \tag{2.50}$$

$$\overline{M}^2 = \frac{V_n^{+2} + V_n^{-2}}{2c_{1/2}^2} \tag{2.51}$$

$$\dot{m} = M_{1/2} c_{1/2} \begin{cases} \rho_L, if M_{1/2} \geq 0 \\ \rho_R, otherwise \end{cases} \tag{2.52}$$

$$M_{1/2} = f_M^+ + f_M^- + M_p \tag{2.53}$$

$$f_M^{\pm} = \begin{cases} \frac{1}{2}(M \pm |M|), if |M| \geq 1 \\ \pm\frac{1}{4}(M \pm 1)^2 \pm \frac{1}{8}(M^2 - 1)^2, otherwise \end{cases} \tag{2.54}$$

$$M_p = -\frac{k_p}{f_a} \max(1 - \sigma \overline{M}^2, 0) \frac{p_R + p_L}{\overline{\rho} c_{1/2}^2} \tag{2.55}$$

$$\overline{\rho} = \frac{\rho_L + \rho_R}{2} \tag{2.56}$$

$$k_u = 0.75, K_p = 0.25, \sigma = 1.0 \tag{2.57}$$

2.5 Time Marching

For time marching a three step Ringe-Kutta scheme, has been applied in the computational model, to achieve an explicit time marching the equation of this method is presented below

$$U^n = L(U) \tag{2.58}$$

$$U^{(1)} = U^n + \Delta t L.(U^n) \tag{2.59}$$

$$U^{(2)} = \frac{3}{4}U^n + \frac{1}{4}U^{(1)} + \frac{1}{4}\Delta t.L(U^{(1)}) \tag{2.60}$$

$$U^{n+1} = \frac{1}{3}U^n + \frac{2}{3}U^{(2)} + \frac{2}{3}\Delta t.L(U^{(2)}) \tag{2.61}$$

2.6 Boundary Conditions

To accomplish CFD computations; a set of boundary conditions have to be implemented such boundary conditions will consists of chamber Inlet, Free stream, Outlet, and Axisymmetric conditions, such boundary conditions is listed below.

Free stream boundary condition:

$$\begin{aligned} \rho_\infty &= 1.0 \\ u_\infty &= 1.0 \\ v_\infty &= 0.0 \\ P_\infty &= 1.0 \end{aligned} \tag{2.62}$$

Inlet boundary conditions:

$$\begin{aligned} \rho_{inlet} &= \rho_{inlet} / \rho_{ref} \\ u_{inlet} &= u_{inlet} / u_{ref} \\ v_{inlet} &= v_{inlet} / v_{ref} \\ P_{inlet} &= P_{inlet} / P_{ref} \end{aligned} \tag{2.63}$$

Outlet boundary condition:

Outlet boundary condition is implemented downstream of the flow; such condition is demonstrated by the equations and figure 2.2.

$$\begin{aligned} \rho_j &= 2\rho_{j-1} - \rho_{j-2} \\ u_j &= 2u_{j-1} - u_{j-2} \\ v_j &= 2v_{j-1} - v_{j-2} \\ P_j &= 2P_{j-1} - P_{j-2} \end{aligned} \tag{2.64}$$

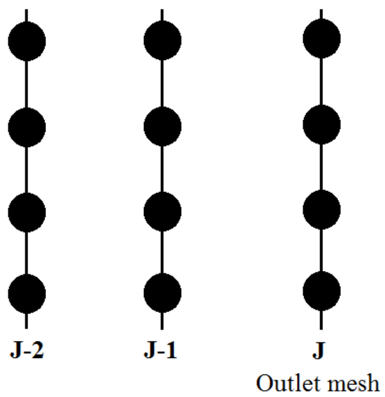


Figure 2.2 Outlet boundary condition

Axisymmetric boundary condition:

Axisymmetric boundary condition is implemented to calculate the flow at the axisymmetric line; such condition is demonstrated by the equations and figure 2.3 below.

$$\begin{aligned} \rho_i &= \rho_{i+1} \\ U_i &= U_{i+1} \\ V_i &= 0.0 \\ P_i &= P_{i+1} \end{aligned} \tag{2.65}$$

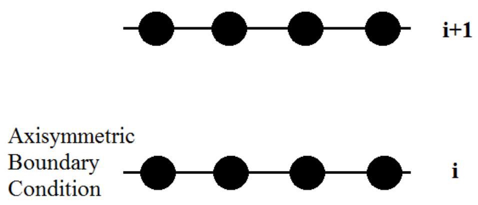


Figure 2.3 Axisymmetric boundary condition

Wall boundary condition:

Wall boundary condition is implemented as a non slip condition; such condition is demonstrated by the equations and figure 2.4 below.

$$\begin{aligned} \rho_i &= \frac{P_{i-1}}{R\,T_w} \\ U_i &= 0.0 \\ V_i &= 0.0 \\ P_i &= P_{i-1} \end{aligned} \tag{2.66}$$

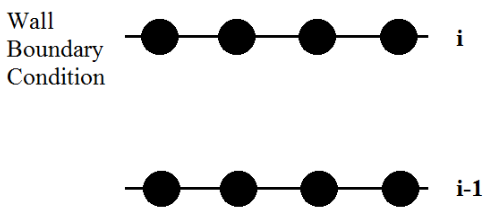


Figure 2.4 Wall boundary condition

Multi zone boundary condition:

To calculate both the internal flow inside the hypersonic body as well as the external flow around it, it is very important to have the capability to compute in a non-rectangular shape regions, the main problem of applying only a rectangular only computational domain is that the CFD solver is only bonded by only an n X m square computational domain:

Figure 2.5 Square Shaped grid

By applying a more than one square grid the CFD user can have a more flexibility making a more complex shape:

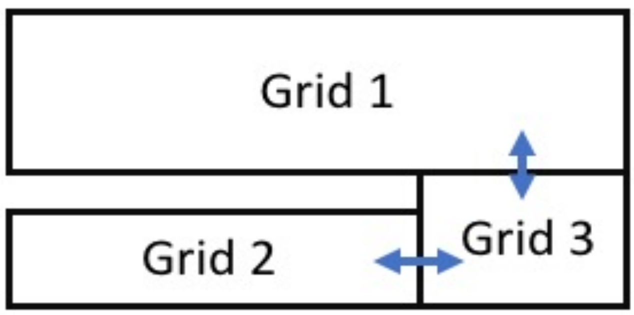


Figure 2.6 Exchange of information

In order to achieve a multi-zone calculation the boundary conditions have to be updated every calculating time step cycle; the coupling between two or more regions is done by implementing a multi zone coupling boundary condition; the equation for coupling two zones is demonstrated by the equations (2.67) and (2.68), along with figure 2.7.

For the first zone:

$$\begin{aligned} q_{zoneA}(1,mx,j) &= q_{zoneB}(1,2,j) \\ q_{zoneA}(2,mx,j) &= q_{zoneB}(2,2,j) \\ q_{zoneA}(3,mx,j) &= q_{zoneB}(3,2,j) \\ q_{zoneA}(4,mx,j) &= q_{zoneB}(4,2,j) \end{aligned} \tag{2.67}$$

For the second zone:

$$\begin{aligned} q_{zoneB}(1,1,j) &= q_{zoneA}(1,mx-1,j) \\ q_{zoneB}(2,1,j) &= q_{zoneA}(2,mx-1,j) \\ q_{zoneB}(3,1,j) &= q_{zoneA}(3,mx-1,j) \\ q_{zoneB}(4,1,j) &= q_{zoneA}(4,mx-1,j) \end{aligned} \tag{2.68}$$

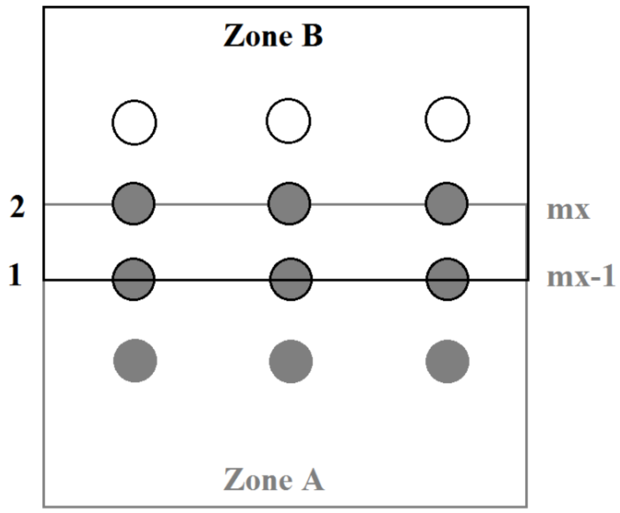


Figure 2.7 Multi zone boundary condition

2.7 Computational Domain

For the numerical results the computational domain was discretized by using two dimensional structured, multi-zone method. The overall domain was divided in six to 4 different zones depending on the operational condition. 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 the body of the hypersonic shape. The final grid size is obtained by performing grid independence study for three coarse, medium and fine size; The minimum mesh size have been used of the order of 1×10-6 m. with the mesh refinement imposed near the walls. The overall computation domain have 250,000 nodes.

Dual jets mesh structure

Computational domain is divided to six different zones, which have two pressure chambers that generates flow that produces coaxial jets.

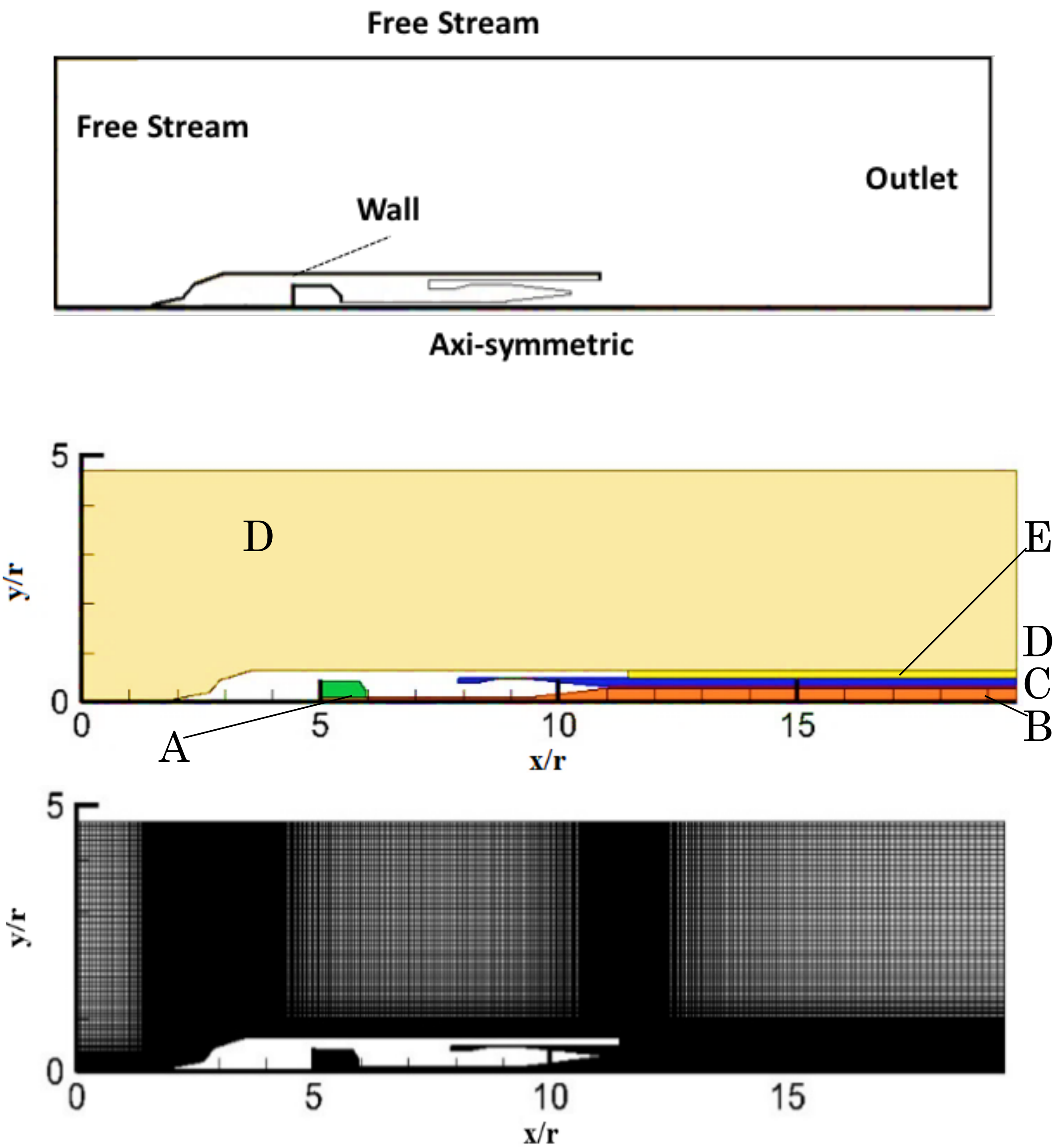


Figure 2.8 Numerical mesh used for duel jet condition

Central jet only in operation mesh structure

Computational domain is divided to four different zones, which have two pressure chambers that generates flow that produces coaxial jets.

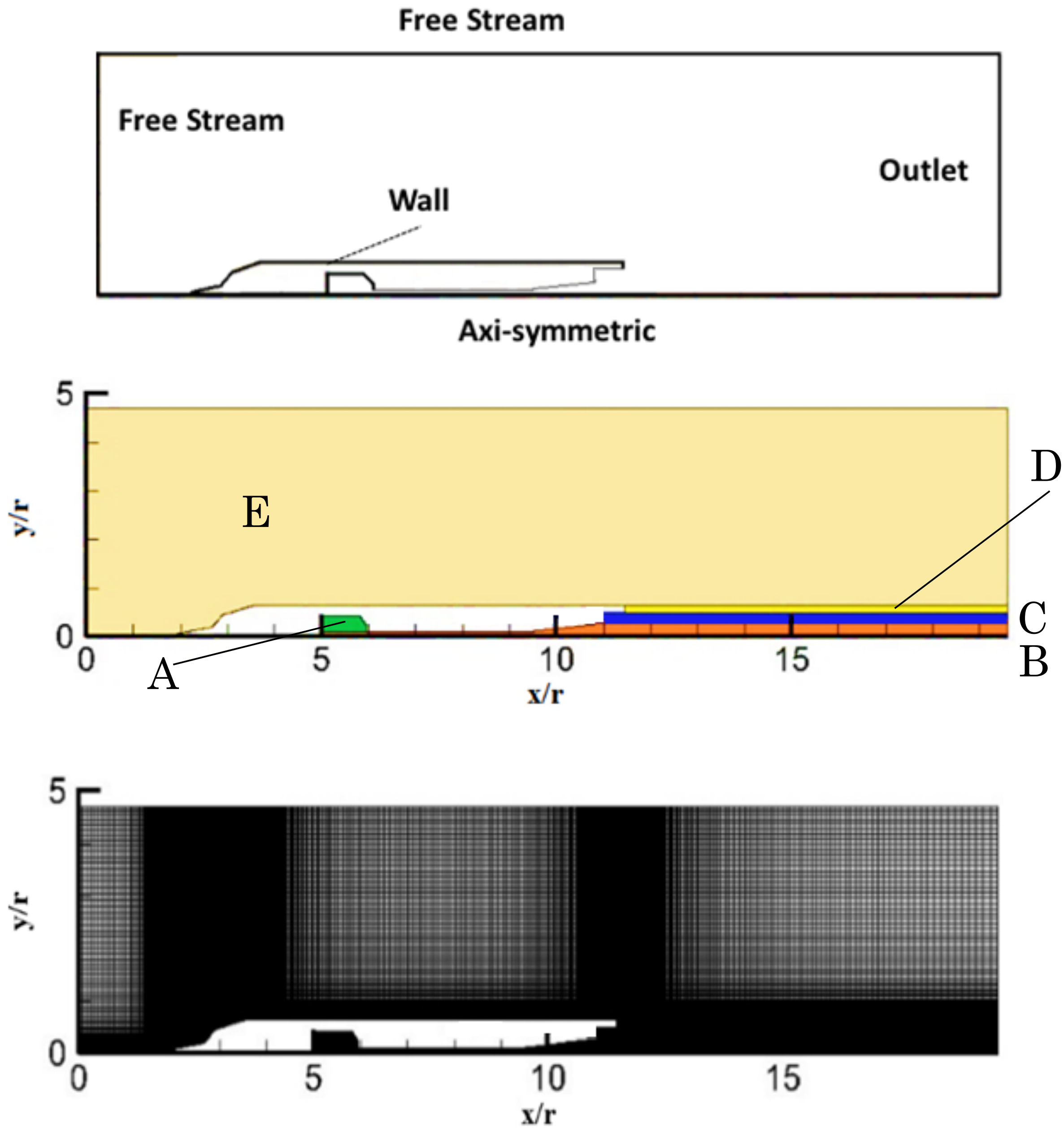


Figure 2.9 Numerical mesh used for central jet condition

Surrounding jet only in operation mesh condition

Computational domain is divided to four different zones, which have two pressure chambers that generates flow that produces coaxial jets.

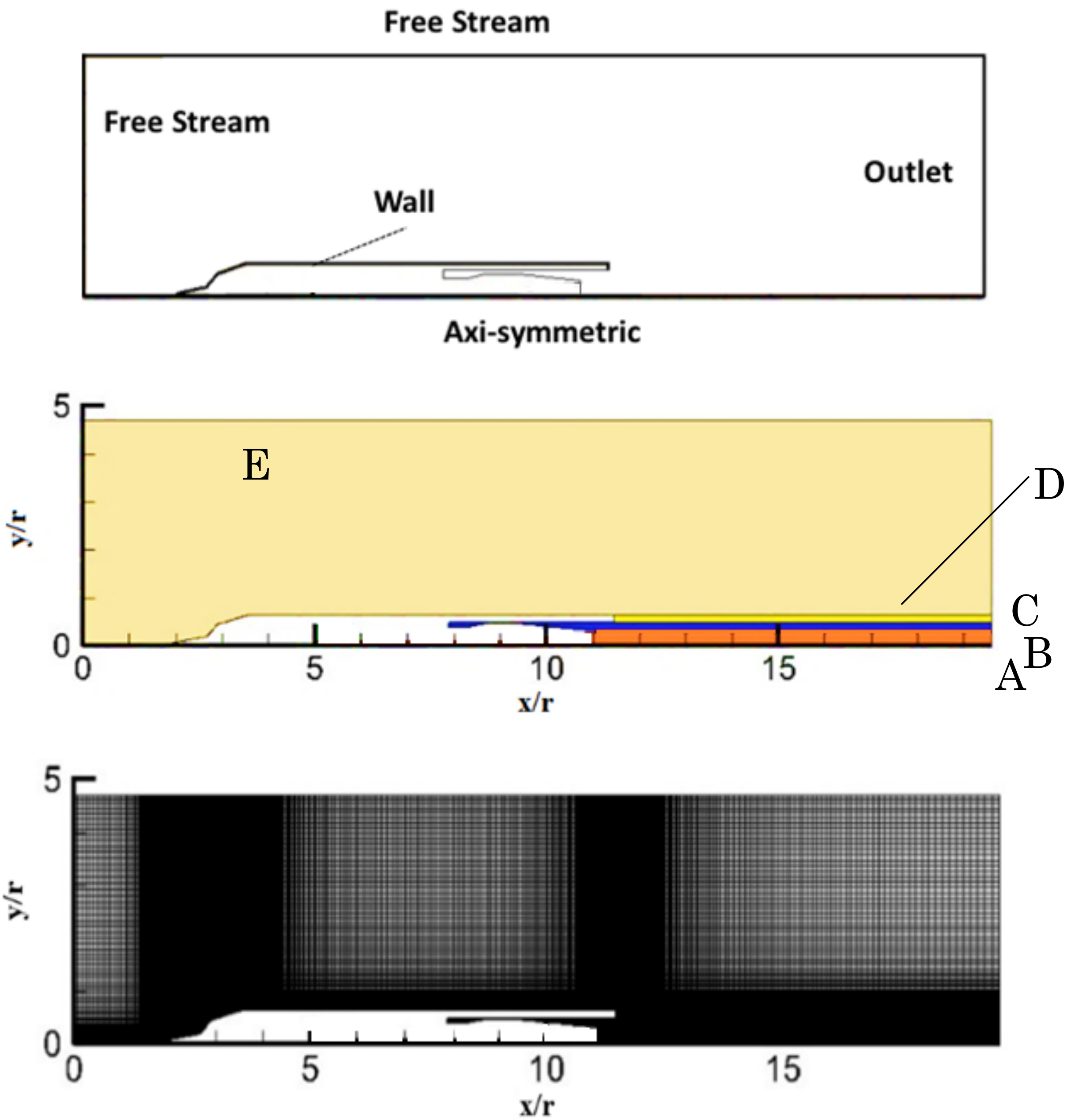


Figure 2.10 Numerical mesh used for surrounding jet condition

Chapter 3

Experimental Methods

Initially, experimental studies were conducted to visualize the basic flow structure in various operation modes. The experiments were conducted in Kashiwa Hypersonic Wind Tunnel Facility at Graduate School of Frontier Sciences, The University of Tokyo Conducting such experiments requires a special experimental set up that matches the University of Tokyo hypersonic wind tunnel [20], the supporting assembly as well as the models has been designed and manufactured in house. The hypersonic models have been supplied by a high-pressured air to be discharged into the hypersonic free stream as a supersonic jet. The Experimental work main objective is to provide the data needed to compare the flow field generated for experiment and the one generated by the CFD results. Two sets of experiments are mainly used, the first is to high resolution images that Capture a static picture and synchronizing it with a full body flow field and a jet focused image, along with record the jets chambers pressures. The second is to provide a More accurate experiment which mainly synchronizes the data gathered from the pressure sensors alongside with high speed camera images.

3.1 Hypersonic Wind Tunnel Facility

3.1.1 Hypersonic Wind Tunnel Specification

Experiments have been conducted at the Kashiwa Hypersonic and High-Temperature Wind Tunnel [20], at the Graduate School of Frontier Sciences, The University of Tokyo shown in figure 3.1. The wind tunnel has a uniform jet core of 120 mm diameter with a Mach 7 flow, maximum stagnation pressure of 950 kPa and maximum stagnation temperature of 1000 K. The general specifications of hypersonic wind tunnel is shown in Table 3.1, For all experiments the average nominal flow stagnation temperature is set to be approximately 600K.

Mach Number	7
Stagnation Pressure	Maximum 0.950 MPa
Stagnation Temperature	Maximum 1000 K
Mass Flow Rate	Maximum 0.39 Kg/sec.
Nozzle Exit	200 mm diameter
Run Time	60 sec. (maximum)

Table3.1 Hypersonic Wind Tunnel General Specifications

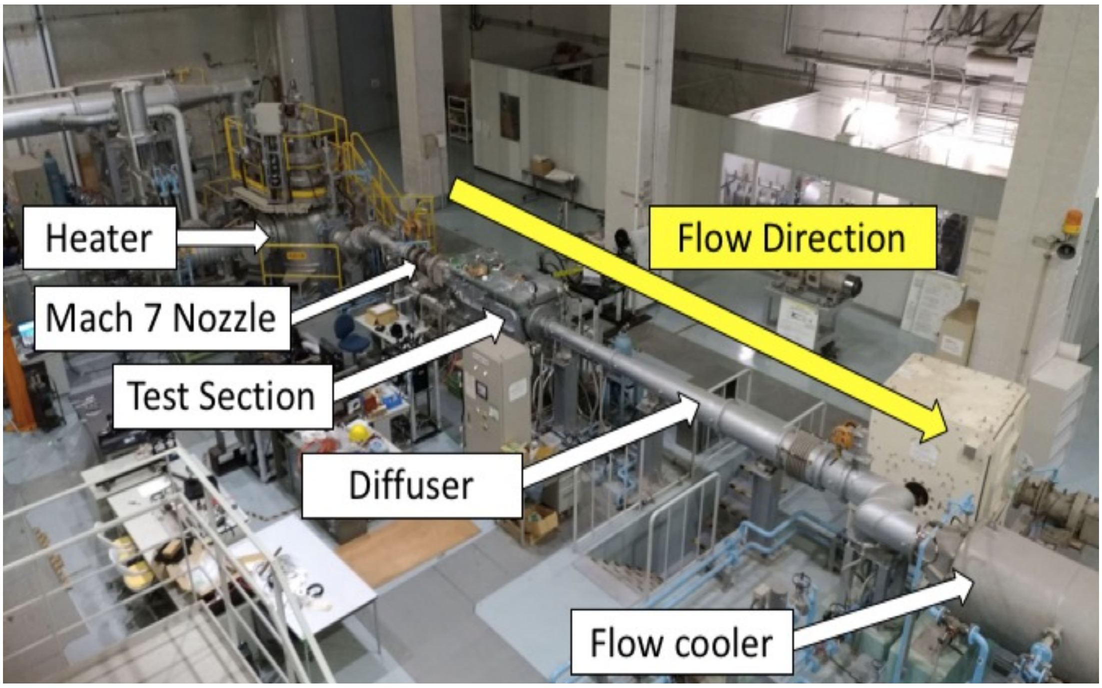


Figure 3.1 Kashiwa Hypersonic Wind Tunnel Facility

For the flow to retch flow velocity of about 1000 m/s, with Mach number of 7, the air flow must be pressurized before the experiment taking place, such process will get the air to a stagnation pressure sufficient to be discharged to a vacuum chamber in which will receive the expanded air at the other end of the wind tunnel, as the flow get expanded from the compressed air chamber to the vacuumed air chamber the flow static pressure drops dramatically, and in order to avoid the problem of air flow liquefaction heat must be added, to the flow to increase the stagnation temperature to a sufficient level that can prevent the flow from being liquefied. For the test section it will be located between the flow heater and the vacuum chamber the flow will be accelerated and decelerated by a nozzle and diffuser located upstream and downstream the test section. The wind tunnel operation sketch as well as the wind tunnel component can be seen in figure 3.2.

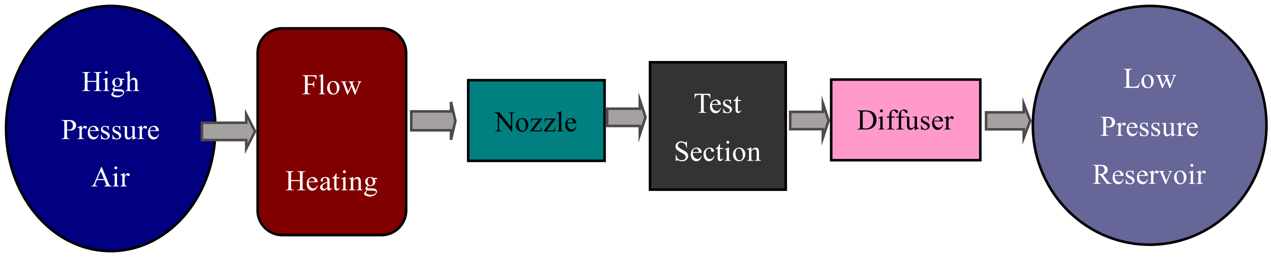


Figure 3.2 University of Tokyo Test Facility Components

3.1.2 Experimental Setup Instrumentation

The experimental method used to simulate a hypersonic engine jet flow is the cold flow injection method, such method requires model small enough to be situated inside the University of Tokyo, kashiwa campus hypersonic wind tunnel test section, such model have to consist of a small settling chamber that can with stand a high pressure that might reach up to 7 atm pressure, take in mind that the jet generator model will be surrounded with a a low pressure environment of around 230 Pa, such vast divergence in the pressure value will impose a high stresses that the model have to withstand. The model is supplied with a high pressure flow feeding line, feeding line will connect the model settling chamber with the high pressure reservoir; the air inside the reservoir will be compressed and stored prior to the experiment, and the feeding flow pressure will be regulated and controlled by a pressure regulator followed by a cut off valve. Pressure is measured for both the supply line as well as the ejected air flow from the jet generator, for the feeding line the pressure will be measured before and after the pressure regulator, a direct measurement of the settling chamber will be recorded. For the ejected jet from the get generator the air jet flow pressure is measured by a three pitot tubes connected to a piezoelectric pressure sensors, and a data collector. The experimental set up general sketch is presented in figure 3.3.

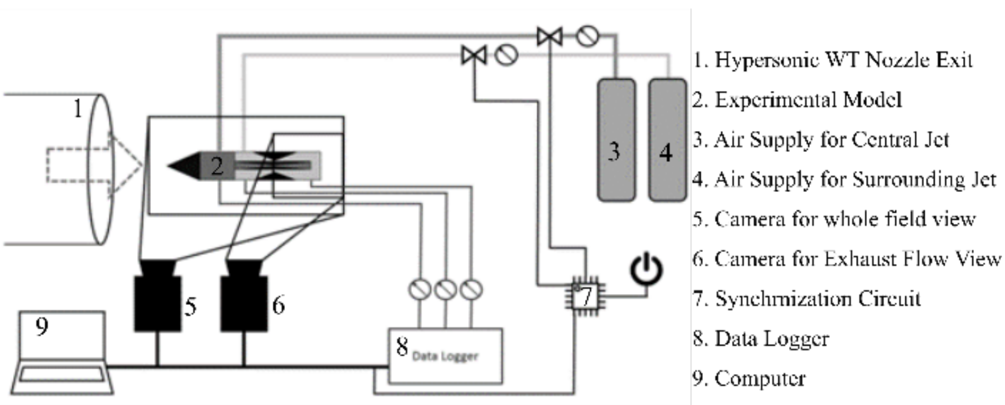


Figure 3.3 Experimental Setup Sketch

Experimental setup is installed in and around the test section of the hypersonic wind tunnel, the test section is shown in figure 3.4, The hypersonic jet generator body is installed through the hypersonic model access hatch, in which is fixed inside the test section this as demonstrated in figure 3.5, and 3.6.

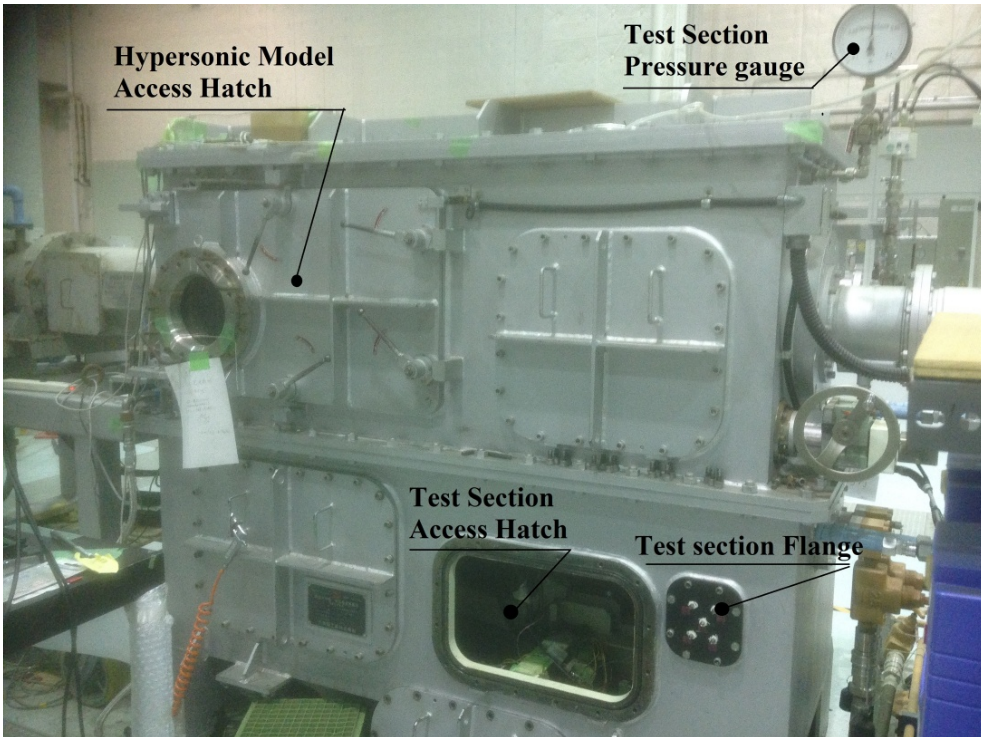


Figure 3.4 Hypersonic Wind Tunnel Test Section

Figure 3.5 Coaxial Jet Generator Inside the Test Section

Figure 3.6   Experimental model Coaxial Jets Nozzle Back View

Design process methodology will be discussed in the following sections in this chapter, however the parts of the jet generator model is presented in figure 3.7.

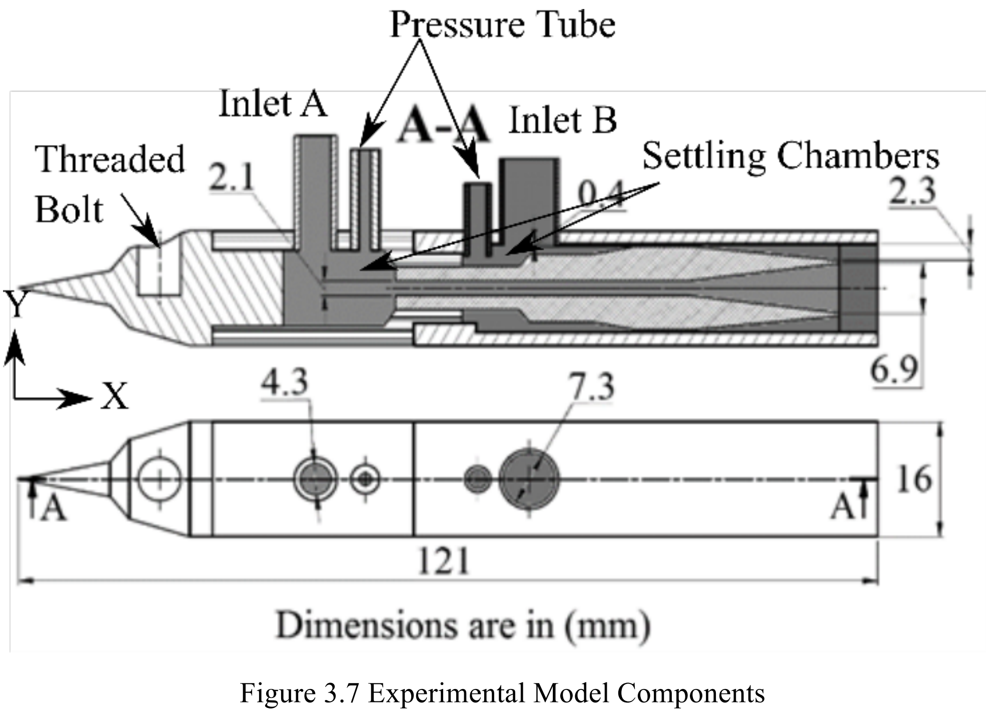


Figure 3.7 Experimental Model Components

Connected to the settling chamber the air supply tube as well as the direct settling chamber measurement tube such tubes are runned from the experimental model through the test section and out of it by passing through the test section flange; this flange will seal the test section from the surrounding much higher pressure around it, as well as allowing the supply tubes to pass air from the high pressure reservoir to the model it also passes the settling chamber measuring pressure tube to the pressure gauge, finally and in regards to the flange mentioned the pressure electrical signal is also passed through this flange to reach the data collector mentioned before, the flange and its components is shown in figure 3.8.

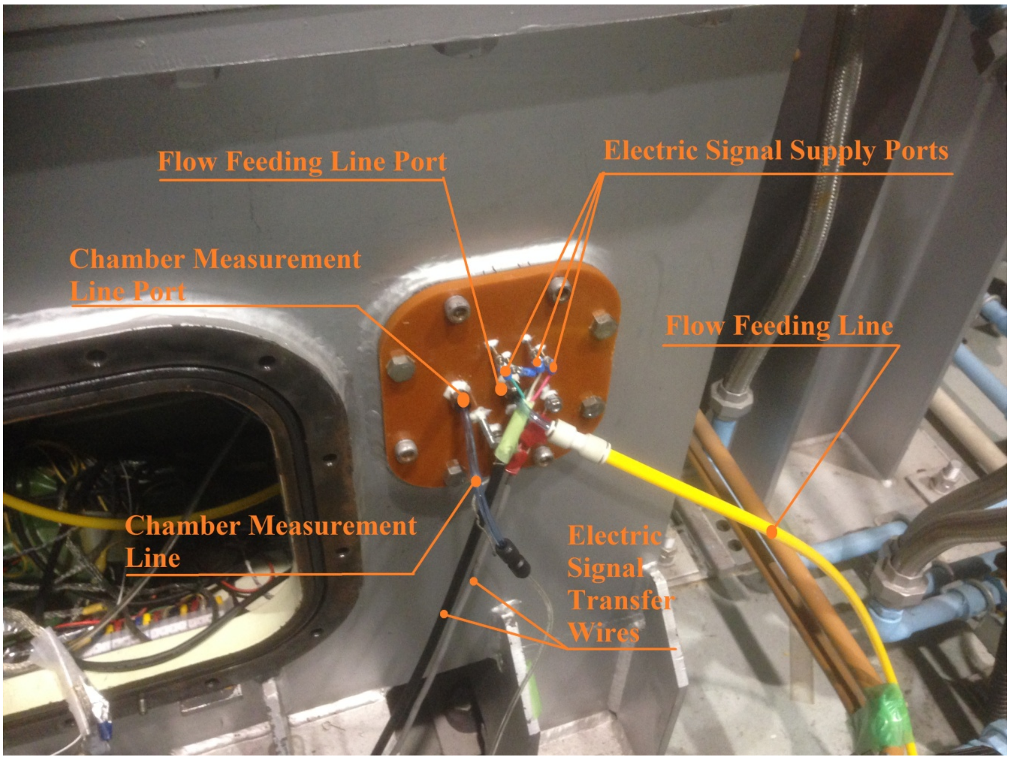


Figure 3.8 Test Section Flange

The jet flow requires a high-pressure flow to be supplied by a compressed air reservoir. data logger connected with flow measurement gauges for the settling chamber, along with the supply reservoir pressure measurement before and after the pressure regulator, the figure also shows the flow cut off valve. In the case of the coaxial jet and for the surrounding jet; it requires a relatively high air mass flow. To visualize the jet flow and compare the general flow field with the CFD data, the hypersonic wind tunnel schlieren system was used which have a light source shown in figure 3.9, and a knife edge as well as two cameras; one will capture the flow field around the whole body including the jet area, (figure 3.10) mean while the other camera is focused on the jet area to acquire a higher accuracy pictures (figure 3.11). A high speed camera is used to synchronize the image with the pressure measurement, the camera used in the current study is the Phantom Miro M310 Camera.

Figure 3.9 Schlieren System Light Source

Figure 3.10 General Flow Camera

Figure 3.11 Jet Focused Camera

Figure 3.12: High Speed Camera (Phantom Miro)

Common channel Pressure measurement port can be seen in the following:

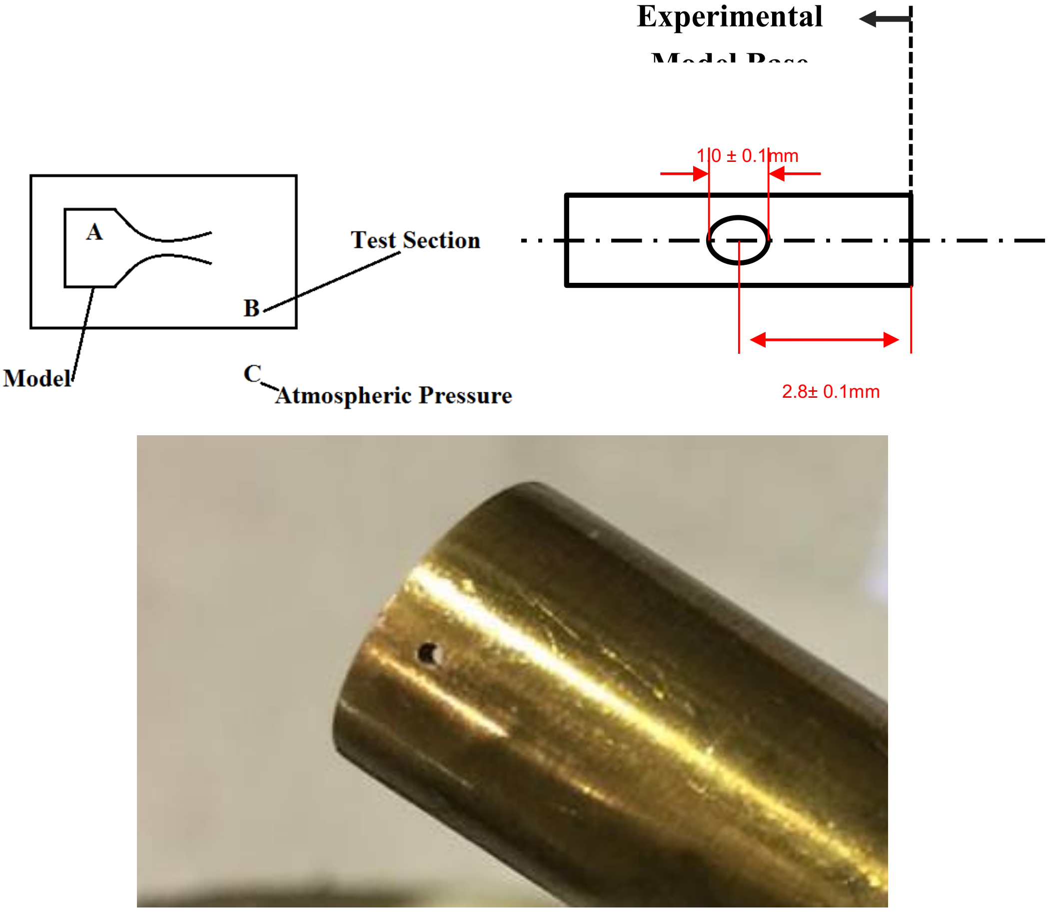


Figure 3.13: Common Channel Pressure Measurement

3.2 Jet Generator Model Design

3.2.1 Isentropic Flow Relations

The Isentropic flow relations have been used to design and predict the performance of the nozzles of coaxial jets, such relation is easy to apply yet can have an acceptable accuracy when predicting the ejected flow out of the nozzle, in the following the Isentropic flow relations been used to design the jet generators nozzles.

$$M = \frac{V}{a} \tag{3.1}$$

$$a = \sqrt{\gamma RT} \tag{3.2}$$

$$P = \rho RT \tag{3.3}$$

$$\frac{P_2}{P_1} = \left(\frac{\rho_2}{\rho_1}\right)^{\gamma}, \; \frac{P_2}{P_1} = \left(\frac{T_2}{T_1}\right)^{\frac{\gamma}{\gamma-1}} \tag{3.4}$$

$$h_0 = h + \frac{1}{2}V^2 \tag{3.5}$$

$$T_0 = T + \frac{1}{2C_p}V^2, \; \frac{T_0}{T} = 1 + \frac{\gamma-1}{2}M^2 \tag{3.6}$$

$$\frac{P_0}{P} = \left(\frac{T_0}{T}\right)^{\frac{\gamma}{\gamma-1}} = \left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}} \tag{3.7}$$

$$\frac{A}{A^*} = \frac{1}{M}\left(\frac{2}{\gamma+1}\right)^{\frac{1}{2}\frac{\gamma+1}{\gamma-1}}\left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{1}{2}\frac{\gamma+1}{\gamma-1}} \tag{3.8}$$

3.2.2 Coaxial jet generator design

Coaxial jet is designed to uncover the effect of coaxial axisymmetric jet on each other; in which have a direct importance in the developing of futuristic propulsion systems; as it represents one of the strongest candidates to propel such aircraft is to be powered by a propulsion system, that has a combined cycle to operate in deferent speed regime.

	Central Jet	Surrounding Jet
Outlet-Throat Area Ratio	10.896	3.1132
Outlet Mach Number	4.0	2.68
Outlet Pressure Ratio	0.0064	0.0445
Outlet Temperature Ratio	0.2364	0.411

Table3.3 Coaxial jet Nozzles isentropic performance

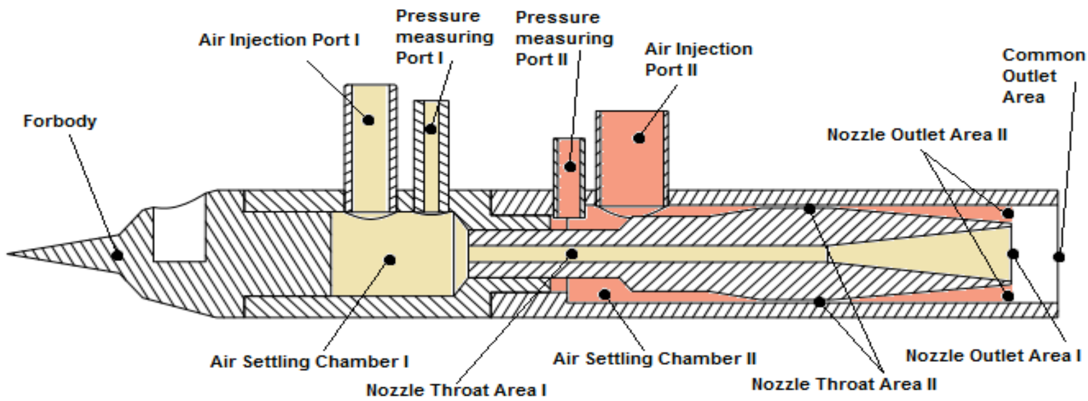


Figure 3.14 Coaxial jet section view

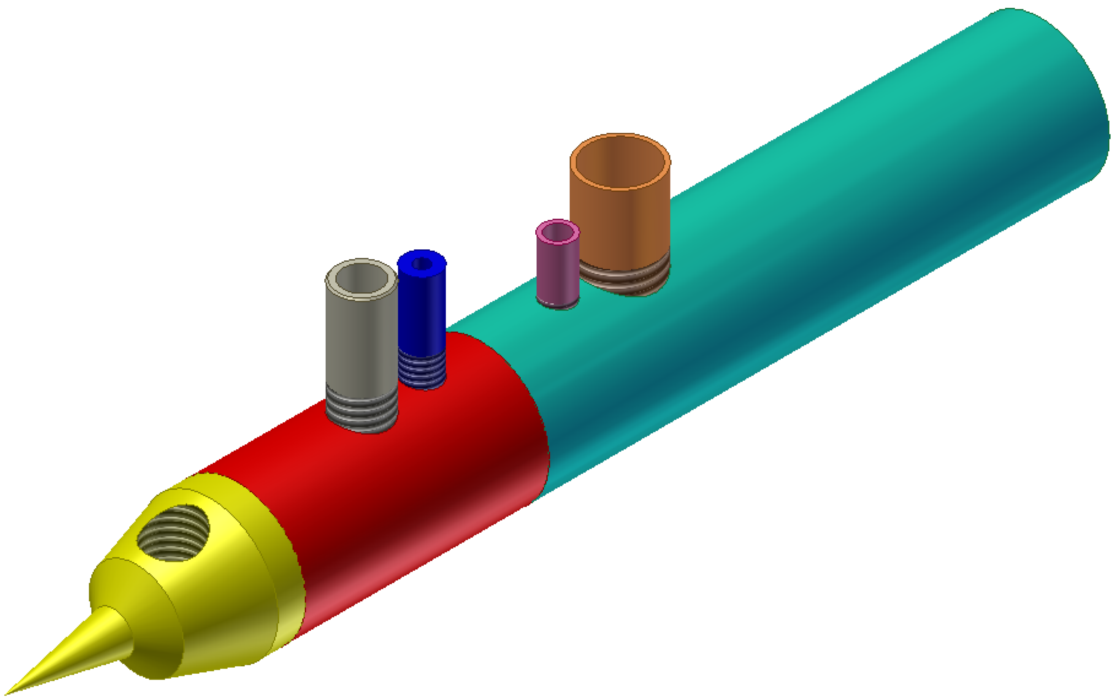


Figure 3.15 Coaxial jet isometric view

3.3 Experimental Setup Manufacturing

As mentioned before the model is designed to be simple enough to be produced in house, by the University of Tokyo Kashiwa campus hypersonic wind tunnel machine shop, the model parts made from brace and have a cylindrical shapes such shapes can be acquired by the use of the lathe machine seen in figure 3.16, Other machine that was very essential in building the test section experimental components is the milling machine seen in figure 3.17, such machine is used to make holes and to create a flat surfaces for the test section model hanging structure.

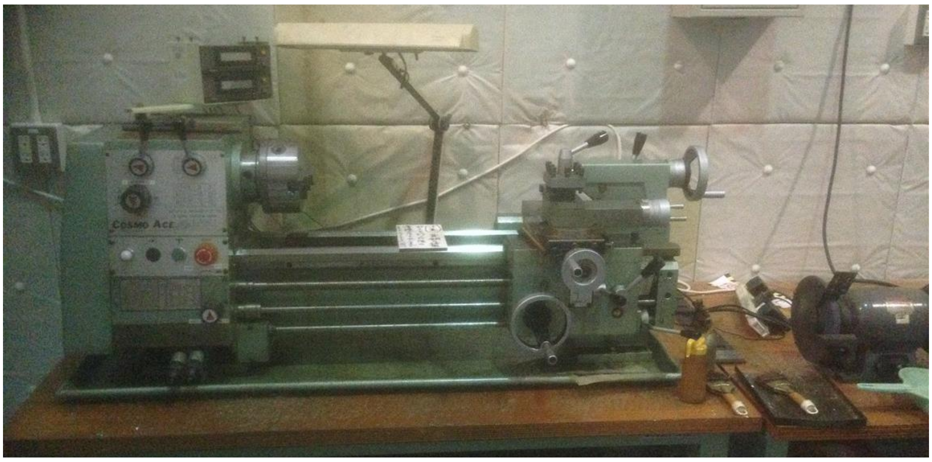


Figure 3.16 Machine shop lathe machine

Figure 3.17 Machine shop milling machine

For the nozzle parts and in order to smooth the internal surfaces of the nozzle a fine grid polish papers has been used. as well as the ture seal adhesive applied on the model supply tube to prevent the flow from leaking out of the model which may cause an undesired flow disturbances as well as a loss in pressure for the model settling pressure chamber. The Coaxial jets model parts can be seen in figure 3.18, The coaxial jets assembly is shown in figure 3.19.

Figure 3.18 Coaxial Jets model parts

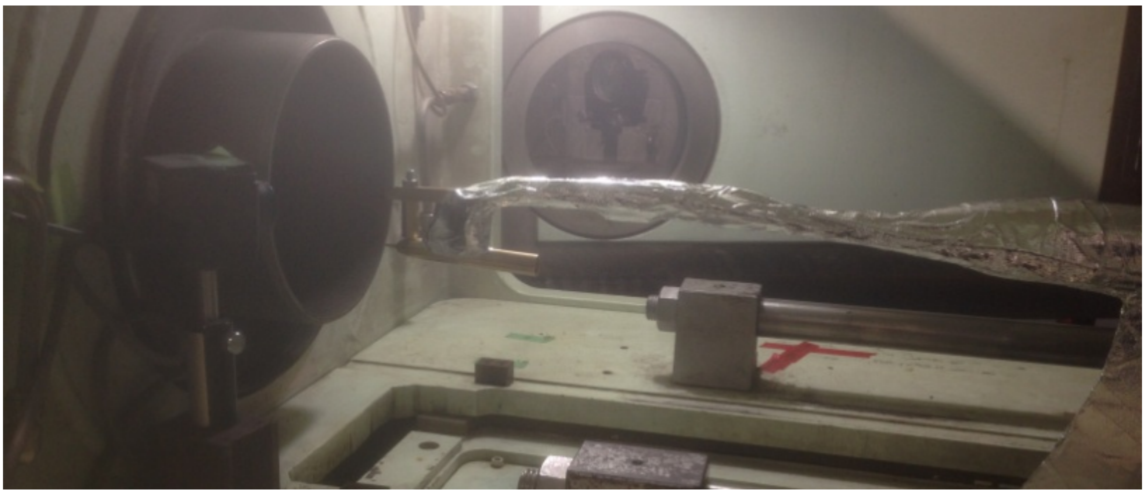


Figure 3.19 Coaxial Jets model Assembly

3.4 Experimental Procedure

3.4.1 Operating Conditions:

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 events during experimental run are 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 process of experimentation can be described in the following figure 3.20.

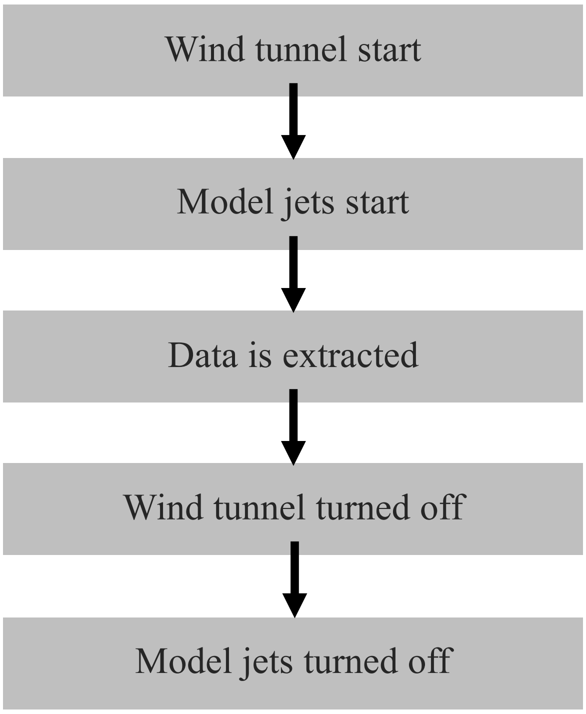


Figure 3.20 Experimental procedure step

The overall run time for the wind tunnel operation was approximately 30 sec. During this run time, all the above-mentioned operations are conducted and pressure and schlieren data were captured. Experiments were conducted by varying settling chamber pressure for central and surrounding nozzle with single common channel length (5.4 mm). However, for the current study only three data point (one for duel operational mode, and two for single operational mode; central jet only and surrounding jet only) has 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. Figure 3.21, and 3.22 shows the pressure measurements for both the jets chambers as well as at the common channel location, note that the spike in the measuring of the common channels are due to some sensor noise.

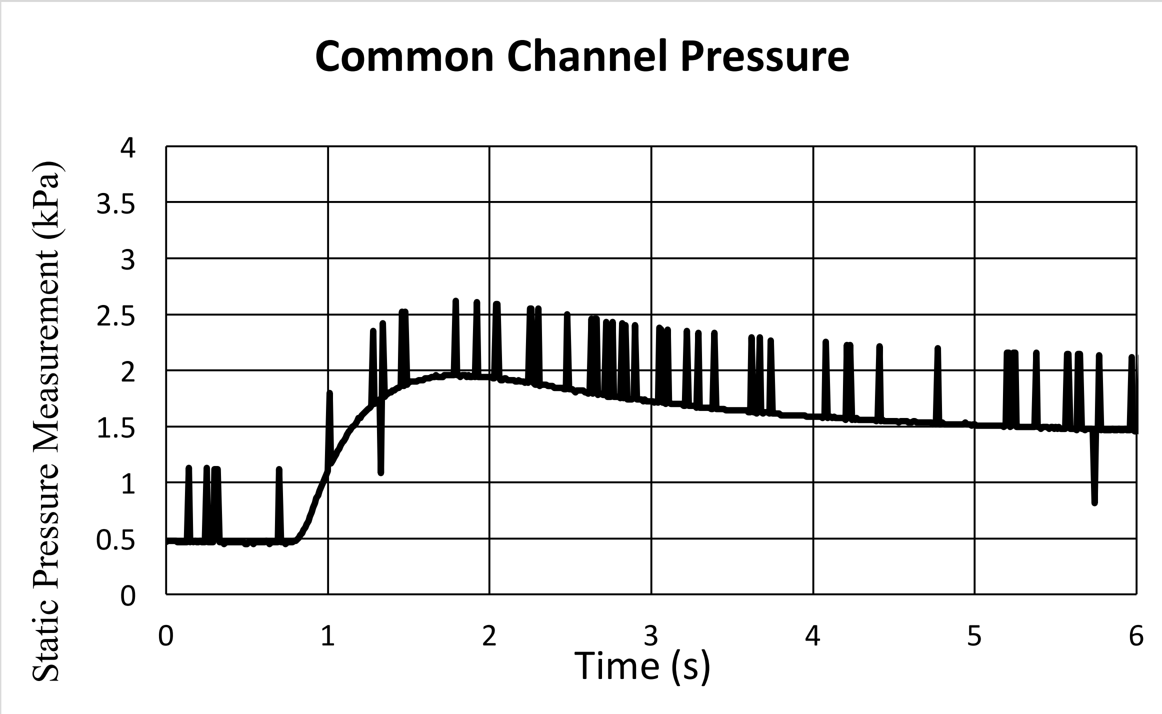


Figure 3.21 Common channel Pressure Measurement

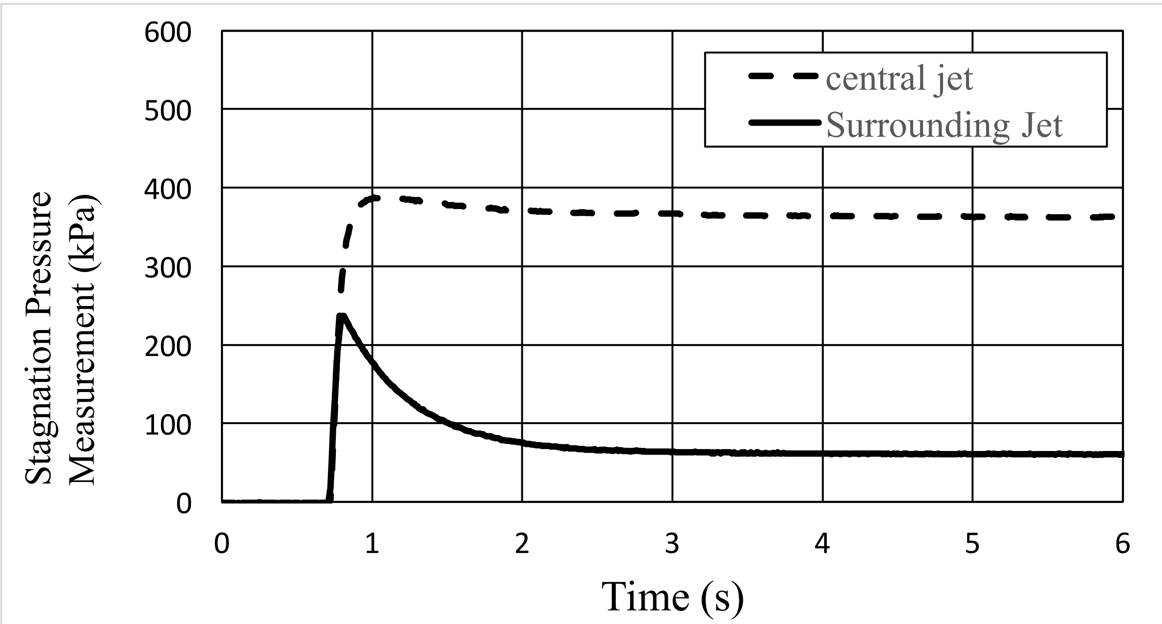


Figure 3.22 Jet Chambers Total Pressure measurement

To achieve the objectives of this research a various types of experiments have been performed such experiments has been summarized by the following:

- No jet flow condition.
- Different chamber pressures.
- Coaxial jet with central jet only on
- Coaxial jet with Surrounding jet only on.
- Coaxial Jets both on.

The data have been acquired by different methods listed below:

- Model colored camera.
- Model schlieren camera.
- Jet focus schlieren camera.
- High Speed camera.
- Pressure Data.

3.7 Test Run and Safety Checks

As seen in the previous discussion the experimental setup uses highly pressurized system to achieve the intended jets flow, that will have some safety issues connected with the use of such high pressure system, such as the failure of the experimental model or it connecting tubes; therefore some responsible safety checks have to be done to ensure the safety of the wind tunnel and its operating staff. The model first tested at low chamber pressures with no hypersonic free stream condition afterwards the system chamber pressure have been raised gradually and the model and its connecting tubes have been checked.

Another issue that can happen that may cause the failure of the experiment is to have a flow break down in the nozzles as a result of its a long and narrow through channels and as seen in figures 3.45, 3.46, and 3.47 the flow was supersonic in all test cases that have been conducted at no hypersonic flow condition, such conclusion is reached by examining the schlieren images generated as a result of these tests, which can be noted the existence of shock train structures within the jet flow of the mentioned models.

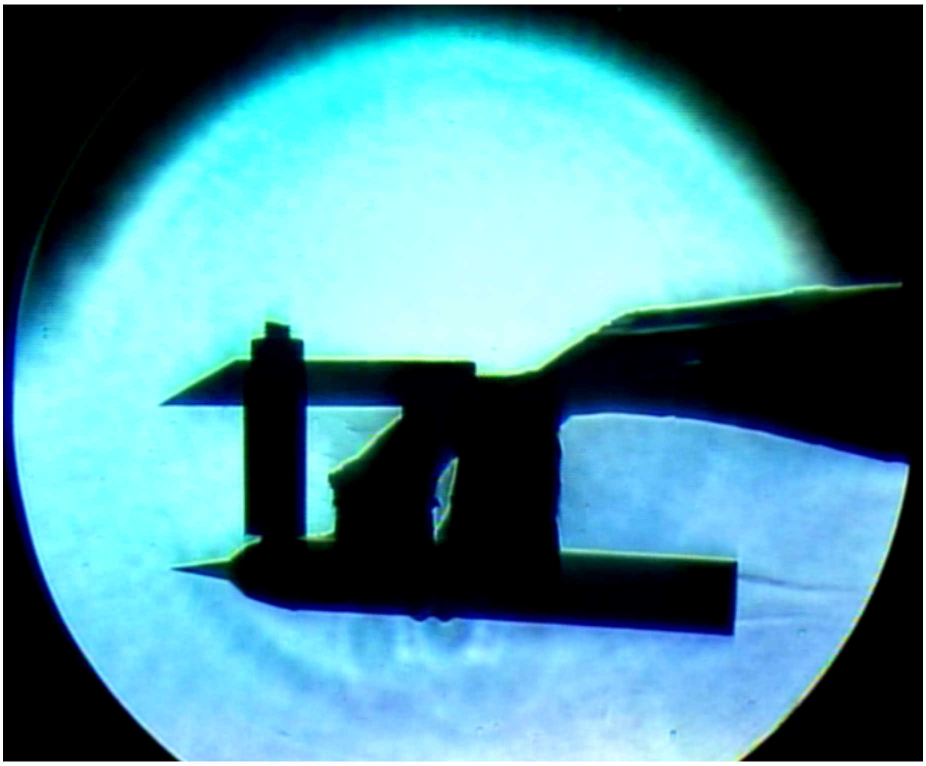


Figure 3.23 Coaxial jet model surrounding jet flow test with no hypersonic flow condition

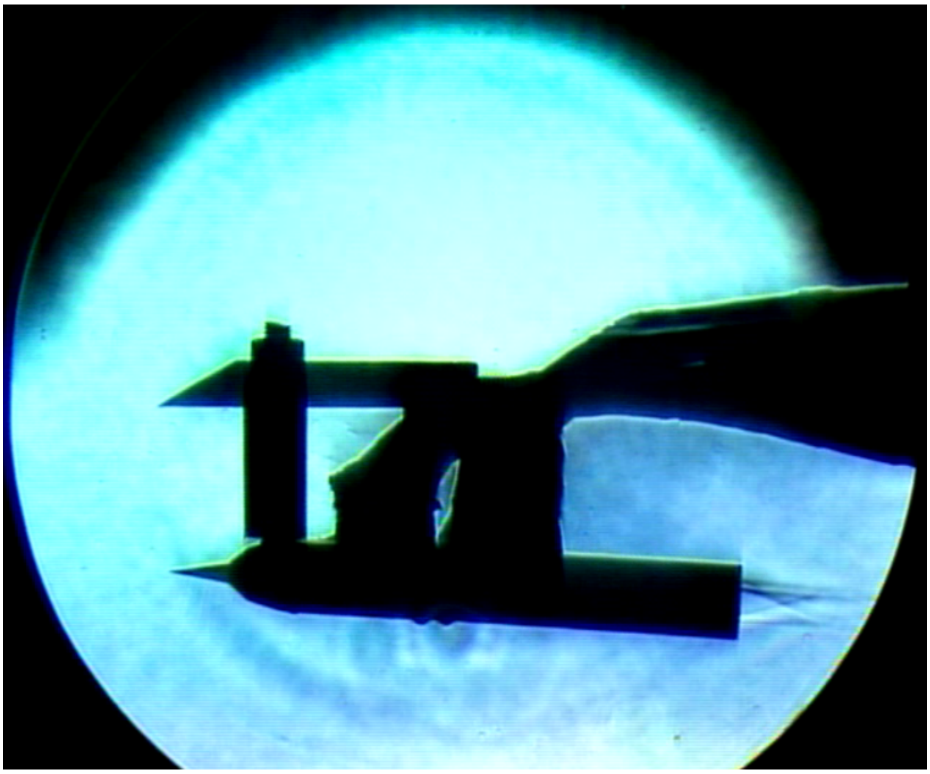


Figure 3.24 Coaxial jet model both jets flow test with no hypersonic flow condition

Chapter 4

Comparison Between Experimental and Numerical Results

Chapter outline:

Experiments are used to evaluate the in-house developed CFD code, which is used to examine and reproduce the qualitative flow features shown by the experiments. Flow features such as the common channel internal oblique shock waves, trailing shock wave, jets boundaries were successfully reproducing. The CFD flow modeling assumes that the flow is laminar. This assumption is reasonable as the Reynolds number of the cold flow (T0=300k) exhausted from the nozzle with a common channel is in the order of 10^4 , also as stated earlier the nozzle wall is manufactured to have a smooth surface. initially the CFD results obtained for dual operation mode qualitatively compared with experimental flow visualization. In the second set of experiments, the wall pressure at 2.8 mm inside from the common channel exit have been measured for various combination of settling chamber stagnation pressure and compared with CFD simulation results.

Note:

This is abridged version of doctoral thesis.

Further contents of Chapter 4 are planned to be part of published Journal articles in future.

The pages 65 – 71 in Chapter 4 will be open to public after March 22nd 2025.

Chapter 5

Coaxial Exhaust Jets Flow Field Analysis

Chapter outline:

The aim of this Chapter is to identify the fundamental characteristics of the interactions between the two streams of the coaxial jets; and to properly assess the interaction between the central and surrounding jet flows and the free stream flow running around the body flying at hypersonic speed. The flow is featured by having two jets, two shear layers, a central Oblique shock wave, and an external trailing shock wave as a result of the interaction between the coaxial jet and the free stream flow around the hypersonic body.
In this section, initially flow features of co-axial supersonic jets in hypersonic environment are discussed for single and dual mode operations for experimental models and corresponding CFD simulations. In the later part, the study of the effect of the common channel on flow field is evaluated by using CFD simulations for the study model that been discussed before.

Note:
This is abridged version of doctoral thesis.
Further contents of Chapter 5 are planned to be part of published Journal articles in future.

The pages 72 – 84 in Chapter 5 will be open to public after March 22nd 2025.

Chapter 6

Common Channel Performance Analysis

Chapter outline:

In this chapter a 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 operating exit Mach numbers on performance in single and dual operation modes have been studied with the thrust evaluation and flow-field analysis. In the last and third part, the effect of addition of various common channel lengths on the flow-field and the thrust performance of the system have been evaluated by numerical simulations.

Note:

This is abridged version of doctoral thesis.

Further contents of Chapter 6 are planned to be part of published Journal articles in future.

The pages 85 – 106 in Chapter 6 will be open to public after March 22nd 2025.

Chapter 7

Conclusion & Future works

Conclusion:

The main motivation of current study was to evaluate performance and flow-field of two coaxial jet system operating in hypersonic environment with the addition of a common channel to the coaxial exhaust flow nozzle. Initially the experimental study have been conducted for small slender body having two high-pressure chamber 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 to evaluate performance and compare flow field with experiments in order to validate CFD results qualitatively. Single point pressure measurement on the wall of common channel was 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 have increased 15-17.4 % with short and long common channel than no common channel case. However, in central jet single mode operation, the thrust have 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 have 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.

As per the current research the use of the common channel is recommended for a coaxial jet with a surrounding jet that have a relatively low exit Mach number with under-expanded jet condition, on the other hand; when having such system of coaxial jets sharing the same exhaust exit area; most of the common area should be utilized by the central jet nozzle; in such descried configuration the common channel will be effective during which the surrounding jet only is in operation. A possible scenario of an effective use of the common channel is when having a coaxial configured RBCC (Rocket Based Combined Cycle) in which the common exit area will mainly used by a central rocket engine that will be active during the subsonic and in space flight regime, and during the hypersonic regime the surrounding jet (can be set as a scram jet) only will be in operation; in which by using the common channel the expansion of the exhaust flow will be improved resulting in an increase in thrust.

Future Works

The current study will evaluate the performance of common channel in following cases: single operation mode with central jet expanding in the common channel, surrounding jet expanding in the common channel, and the effect of the common channel when both jets injected in the common channel. The main objective of current study is to evaluate the performance of co-axial nozzle in hypersonic environment and study the effect of the common channel to enable thrust augmentation for various operation modes. The study is conducted by qualitative experimental flow visualization of shock structure for single and dual operation modes with common channel. Further, numerical simulations are performed for cases to evaluate the performance of system by thrust calculations. Further parametric study has been performed for resized model by isentropic as well as CFD simulations for various central and surrounding jet Mach numbers without common channel. Further, CFD simulations have been performed to study the effect of common channel on flow field and thrust performance.

As per the current research the use of the common channel is recommended for a coaxial jet with a surrounding jet that have a relatively low exit Mach number with under-expanded jet condition, on the other hand; when having such system of coaxial jets sharing the same exhaust exit area; most of the common area should be utilized by the central jet nozzle; in such descried configuration the common channel will be effective during which the surrounding jet only is in operation. A possible scenario of an effective use of the common channel is when having a coaxial configured RBCC (Rocket Based Combined Cycle) in which the common exit area will mainly used by a central rocket engine that will be active during the subsonic and in space flight regime, and during the hypersonic regime the surrounding jet (can be set as a scram jet) only will be in operation; in which by using the common channel the expansion of the exhaust flow will be improved resulting in an increase in thrust.

For the future work, it is recommended to study the process of coaxial jets flows starting; such starting process can have an interesting behavior of jets as it get through from starting phase to steady state phase of operation; it is interesting to investigate the starting sequence of the coaxial jets system and how it will affect the flow structure during the stating phase. For the common channel it is also interesting to study the effect of the common channel as it operates through different free stream Mach numbers from subsonic, supersonic to hypersonic flow; and the effect of the free stream pressure on flow structure and performance of the coaxial jets system common channel.

References

[1] www.stanford.edu ,http://www.stanford.edu/group/fpc/cgi-bin/fpcwiki/Main/Research.

[2] Aristides M. Bonanos, Joseph A. Schetz, Chair Walter F. O'Brien Christopher P. Goyne Richard W. Barnwell Joseph J. Wang, Scramjet Operability Range Studies of an Integrated Aerodynamic-Ramp-Injector/Plasma-Torch Igniter with Hydrogen and Hydrocarbon Fuels, Dissertation submitted to the Faculty of the Virginia Polytechnic Institute and State University In partial fulfillment of the requirements for the degree of Doctor of Philosophy In Aerospace Engineering, August 2005.

[3] K. Takayuki, and T. Nobuhiro, Development Study on Air Turbo-Ramjet for Space Plane, Institute of Space and Astronautical Science, 2003.

[4] H. Taguchi, K. Harada, H. Kobayashi, T. Kojima, M. Hongoh, D. Masaki, S. Sawai, Y. Maru, and T. Sato, Firing Test of a Hypersonic Turbojet Engine Installed on a Flight Test Vehicle, American Institute of Aeronautics and Astronautics, 2009-7311,2009.

[5] http://www.reactionengines.co.uk/sabre_howworks.html.

[6] Rathakrishnan E.: Applied Gas Dynamics,1st ed. ,John Wiley & Sons Singapore, 2010, pp 505-509.

[7] T.A. Edwards. (1989). Numerical Investigation of Hypersonic exhaust plume/afterbody flow field. Tenth Australian Fluid Mechanics Conference- University of Melbourne.

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

[9] K.M.Pandey, Virendra Kumar.(2010). CFD Analysis of Twin Jet Flow At Mach 1.74.with Fluent Software, International Journal of Environmental Science and Development, 2010-0264.

[10] Ramesha D.K., Rudra Murthy HemanthKumar.P.: CFD Analysis of Supersonic Exhaust in a Scramjet Engine, International Journal of Innovative Research in Science, Engineering and Technology (An ISO 3297: 2007 Certified Organization) Vol. 2, Issue 9, September 2013.

[11] Peter W. Merlin: Design and Development of the Blackbird: Challenges and Lessons Learned, AIAA 2009-1522, January 2009 pp 22-27.

[12] Rupesh Aggarwa1 ,KhushinLakhara , P.B. Sharma , Tocky Darang , Naman Jain , Siddharth Gangly: SABRE ENGINE: Single Stage to Orbit Rocket Engine, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 4, Issue 10, October 2015, pp 10360 -10366.

[13] Sharma, H., Vashishtha, A., Rathakrishnan, E., &Lovaraju, P. (2008). Experimental study of overexpanded co-flowing jets. The Aeronautical Journal (1968),112(1135), 537-546. doi:10.1017/S0001924000002499.

[14] U. Mehta, M.Aftosmis, J. Bowles, and S. Pandya: Skylon Aerodynamics and SABRE Plumes, American Institute of Aeronautics and Astronautics (AIAA), 2015. 2015-3605, pp 1-21.

[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] M.S. Liou, A sequel to AUSM, Part II: AUSM+-up for all speeds, Journal of Computational Physics, Vol. 214, pp 137–170, 2006.

[18] B. van Leer, Upwind and High-Resolution Methods for Compressible Flow: From Donor Cell to Residual Distribution Schemes, Communications in Computational Physics Vol. 1, No. 2, pp. 192-206 , 2006.

[19] S. Gottlieb, Journal of Scientific Computing, Vol. 25, Nos. 1/2, 2005.

[20] Imamura, O., Watanuki, T., Suzuki, K. and 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.

[21] Kenneth E. Tatum, Lawrence D. Huebner.: Exhaust Gas Modeling Effects on Hypersonic Powered Simulation at Mach 10, American Institute of Aeronautics and Astronautics (AIAA),1995. 95-6068.

[22] S.Crist, P.Sherman, and D. Glass, 'Study of the Highly Underexpanded Sonic Jet', AIAA Journal Vol.4, No. 1, March, 1965.

[23] I.Chang, and W.Chow, 'Mach Disk from Underexpanded Axisymmetric Nozzle Flow', AIAA Journal Vol.12, No.8 August 1974.

[24] C.Tam, and H.Tanna, 'Shock Associated Noise of Supersonic Jets From Convergent-Divergent Nozzles', Journal of Sound and Vibration in a paper published,1982.

[25] J. Shirie, and J. Seubold, 'Length of the Supersonic Core in High Speed Jets', AIAA Vol.5 No, 11, 1967.

[26] S.Chuech, M. Lai, and G. Faeth, 'Structure of Turbulent Sonic Underexpanded Free Jets', AIAA Col.27, No. 5 1989.

[27] H.Korst, R. White, S.Nyberg, and J.Agrell, 'The Simulation and Modeling of Jet Plumes in Wind Tunnel Facilities', 11th Aerodynamic Testing Conference, March 1980, Colorado Springs, USA.

[28] S.Nyberg, J. Agrell, and T. Heverng, 'Investigation of Modeling Concepts For Plume-Afterbody Flow Interactions' 2nd Annual Technical Report with the title of 1980.

[29] H.Korst, and R. White, 'Simulation and Modeling of Jet Plumes in Wind Tunnel Facilities', AIAA Vol. 18, No.5, September 1981.

[30] G. Deiwert, 'A Computational Investigation of Supersonic Axisymetric Flow Over Boattails Containing a Centered Propulsive Jet', 21st Aerospace Sciences Meeting, January 1983.

[31] G. Deiwert, 'Supersonic Axisymetric Flow Over Boattails Containing a Centered Propulsive Jet', AIAA Vol.22, No.10 1984.

[32] J. Sahu, 'Computations of Supersonic Flow Over a Missile Afterbody Containing an Exhaust Jet', Journal of Spacecraft (JSR), VOL 24, No. 5, 1987.

[33] M. Dash, N.Sinha ‘Exhust Plumes and their Interaction With Missile Airframes- A New Viewpoint’ NASA STI/Recon Technical Report A. January 1992.

[34] S.Svirschevsky, N. Semenchikov, N. Korolev ‘Research of Aerodynamics of Misslile Afterbody with Ring Nozzle in Trans and Supersonic Stream’, Mathematical Modeling, 1999.

[35] P.Lovaraju, E. Rathankrishnan, ‘Experimental Studies on Co-Flowing Subsonic and Sonic Jets’, Flow, Turbulence and Combustion, Vol. 87, Issue 1, July 2011.

[36] V. Ivanov, S. Frolov, V. Zvegintesv, V. Aksenov, I. Shamshin, D. Vnuchkov, D. Nalivichenko, A. Berlin, V. Fomin, A. Shiplyuk, and N. Yakovelv ‘Hydrogen- Fueled Detonation Ramjet Model: Wind Tunnel Tests’ AIP Conference Proceedings, November 2018.

[37] S. Ashfaq, and S. Khan, ‘Combined Effect of Relief and Level of Expansion in a Suddenly Expanded Flow’, Journal of Mechanical and Civil Engineering (IOSR-JMCE), VOL. 12, Issue.5, Sep – Oct 2015.

[38] 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.

[39] 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.

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

[41] 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.

[42] 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.

[43] 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.

[44] 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.

[45] 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).

[46] http://www.skytamer.com/Pratt&Whitney_JT11.html.

[47] http://www.sr-71.org/blackbird/manual/.

[48] http://www.rocket.jaxa.jp/fstrc/0a06_06.html.

[49] Derek J. Dalle_, Sean M. Torrez_, James F. Driscolly, .(2010). Reduced-Order Modeling of Reacting Supersonic Flows in Scramjet Nozzles, American Institute of Aeronautics and Astronautics, 2010-6958.

[50] http://www.cfd-online.com/Wiki/Isentropic_flow_relations.

[51] H. B. Hopkins, W. Konopka, J. Leng, Validation of Scramjet Exhaust Simulation Technique, NATIONAL AERONAUTICS AND SPACE ADMINISTRATION, WASHINGTON, D. C. JUNE 1976.

[52] Advisory Group for Aerospace Research and Development (AGARD), Hypersonic Combined Cycle Propulsion, Hypersonic Combined Cycle Propulsion Panel Symposium (75th) Held in Madrid, Spain on 28 May - I June 1990 (La Propulsion Hypersonique a Cycles Combines.

[53] R. A. Baurle, J. R. Edwards, Hybrid Reynolds-Averaged/Large-Eddy Simulations of a Coaxial Supersonic Free-Jet Experiment, 47th AIAA Aerospace Sciences Meeting Including The New Horizons Forum and Aerospace Exposition 5 - 8 January 2009, Orlando, Florida, AIAA 2009-129.

[54] professor Suzuki, Doctor Watanaba teaching material for Basic CFD - Intuitive understanding 'Exercise 6 May 27, 2014'.

[55] www.edx.org, Hypersonics – from Shock Waves to Scramjets Course.

[56] professor Kojiro Suzuki lectures note ' ntroduction to Computational Fluid Dynamics'.

[57] Pei Chen, Suxun Li, Shijie Luo, ZhaoyongNi.(2014). Flow Visualization on Lateral Multiple Jet Interaction with Freestream, China Academy of Aerospace Aerodynamics (CAAA).Journal of Flow Control, Measurement & Visualization, 2014, 2, 7-11.

[58] www.jaxa.jp , http://www.jaxa.jp/article/special/aviation/ishikawa02_j.html.

[59] S. Stephan and R. Radespiel, R. Müller-Eigner, PROPULSIVE JET SIMULATION IN A HYPERSONIC LUDWIEG TUNNEL, German Aerospace Congress 2012, Berlin, German Society for Aerospace - Lilienthal-Oberth eV, Bonn, 2012.

[60] Christopher Matthew Roseberry, Donald R. Wilson, ARC-HEATED GAS FLOW EXPERIMENTS FOR HYPERSONIC PROPULSION APPLICATIONS, doctoral dissertation of Christopher Matthew Roseberry, 2005.

[61] Corin Segal, The Scramjet Engine PROCESSES AND CHARACTERISTICS, University of Florida, 2009.

[62] Mohamed A. Al Kady and Farouk M. Owis, NUMERICAL SIMULATION OF NOZZLE FLOW WITH CHEMICAL EQUILIBRIUM, ARPN Journal of Engineering and Applied Sciences, ISSN 1819-6608, VOL. 8, NO. 5, MAY 2013.

[63] A.Ciucci, Development of Numerical Model of the Nozzle Flow in Law Power Arcjet Thrusters, International Electric Propulsion Conferance IEPC-93-182 September-1993.

[64] M. J. Kermani, E. G. Plett, Modified Entropy Correction Formula for the Roe Scheme, American Institute of Aeronautics and Astronautics Paper 2001-0083.

[65] RabahHaoui, Effect of Mesh Size on the Supersonic Viscous Flow Parameters around an Axisymmetric Blunt Body, World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering Vol:8,

No:7, 2014.
[66] Erinc Erdogan, Development of Navier-Stokes Solver for Multi-Block Applications, A thesis submitted to the Graduat School of Natural and Applied Sciences of Middle East Technical University, September- 2004.

[67] A. D. Cutler, J. A. White, AN EXPERIMENTAL AND CFD STUDY OF A SUPERSONIC COAXIAL JET, American Institute of Aeronautics and Astronautics AIAA-2001-0143.

[68] Meng-Sing Liou, The Evolution of AUSM Schemes, Defence Science Journal, Vol. 60, No. 6, pp. 606-613, November 2010.

[69] Keiichi Kitamura, EijiShima, A New Pressure Flux for AUSM-Family Schemes: Toward Shock-Stable, All-Speed Flux Functions for Hypersonic Heating, Japan Society of Fluid Mechanics (JSFM) B12-3, 2010.

[70] Kojima Takayuki, and TanatsuguNobuhiro, Development Study on Air Turbo-Ramjet for Space Plane, Bulletin of GTSJ 2003.

[71] Kwon-Hee Lee, STUDY OF SUPERSONIC, DUAL, COAXIAL, SWIRLING JET, A dissertation submitted for the degree of Doctor of Engineering, September 2004.

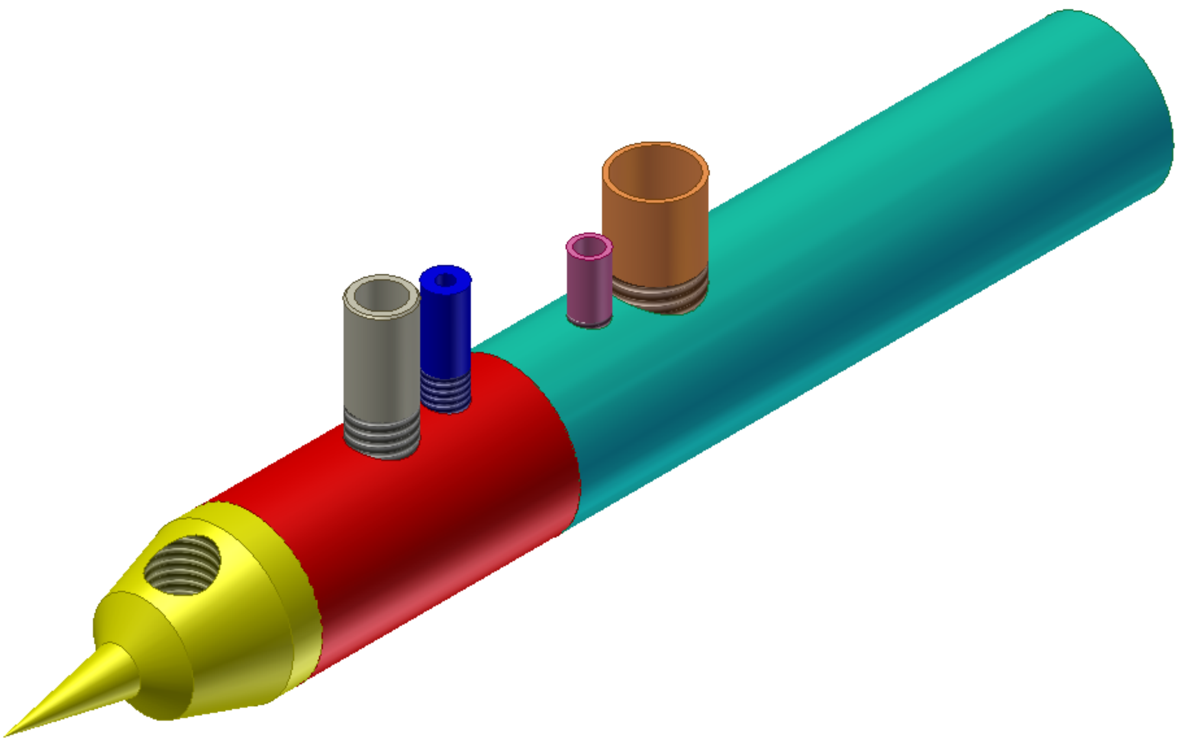
[72] Mohammad Samara, Numerical and Experimental Investigation on Interaction between Flow around a Hypersonic Body and Supersonic Jet from its Tail, 30th (ISTS) International Symposium on Space Technology and Science conference, July 4-10. 2015.

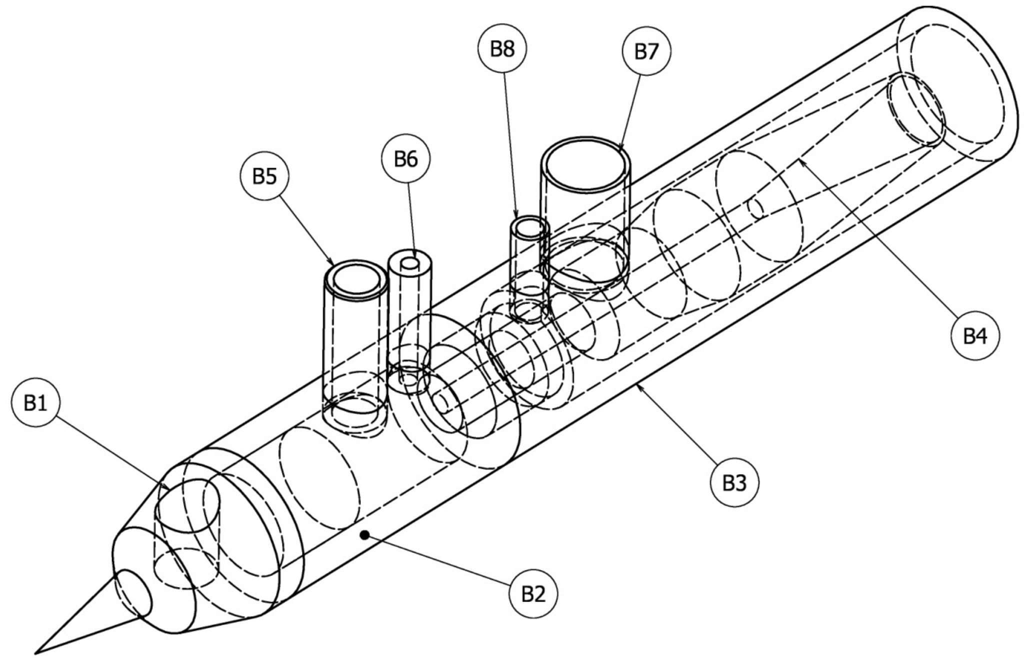
[73] Mohammad Samara, Yasumasa Watanabe, Kojiro Suzuki, Experimental and Numerical Investigation on Coaxial Jets Exhausted from a Body in Hypersonic Flow, 13th International Symposium on Fluid Control,FLUCOME2015/ Measurement and Visualization, , Doha, Qatar, November 15-18, 2015.

Appendix I

Coaxial Jets Experimental Model Design

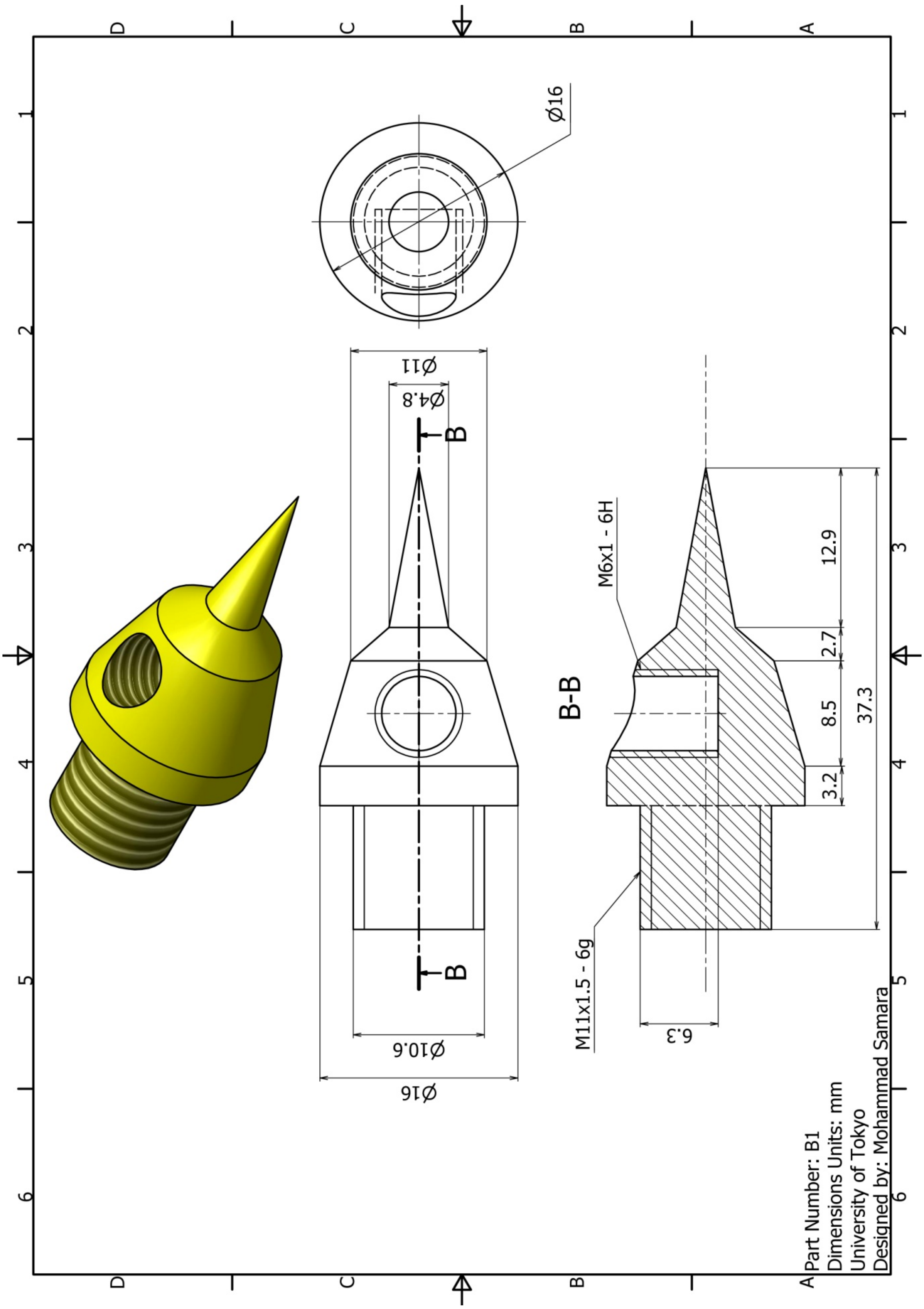
The following will present the detailed design of the coaxial jets Experimental model.



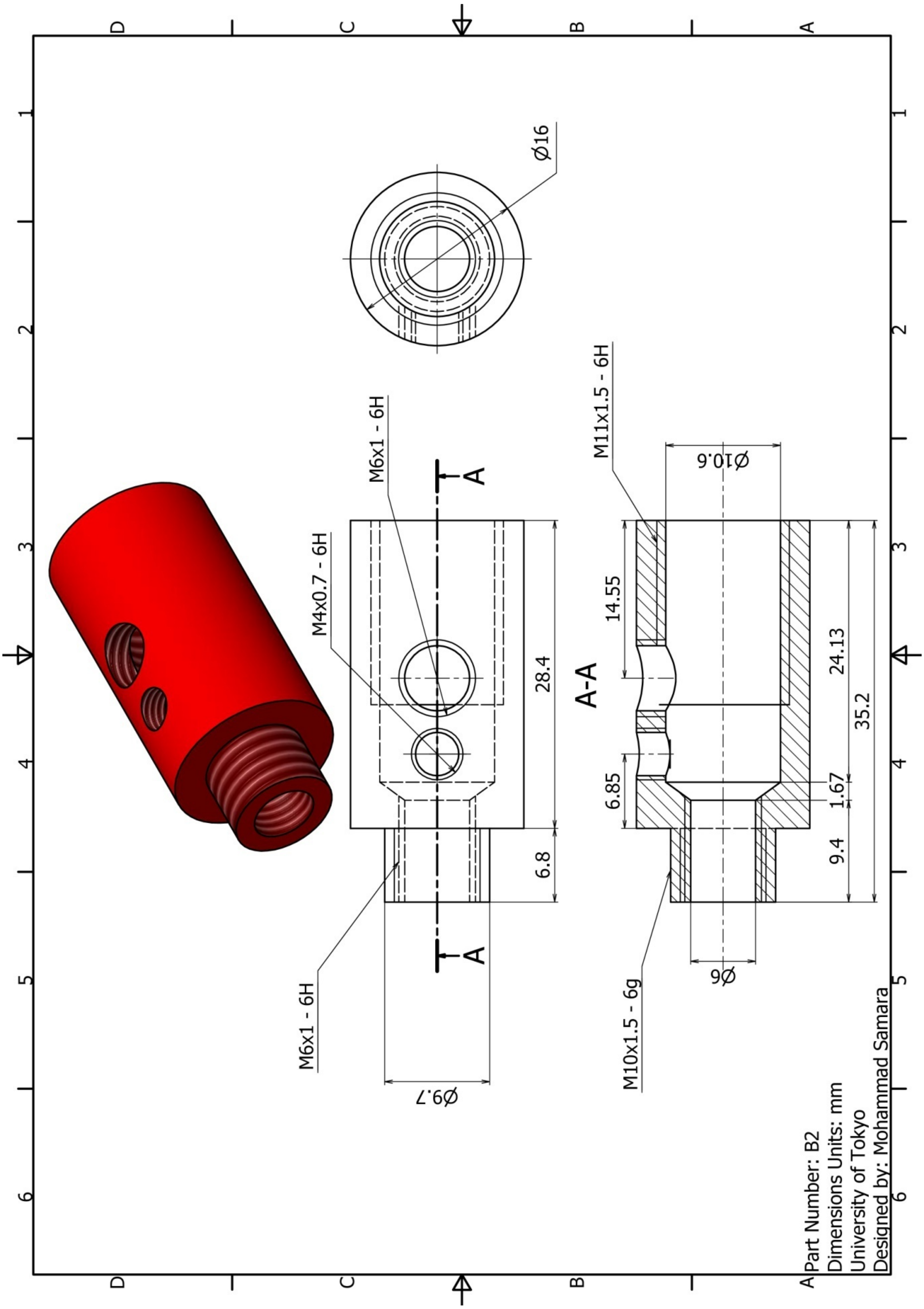


Coaxial Jet Model		
Part Number	Part Name	Parts PerAssembly
B1	Nose cone	1
B2	Central jet chamber	1
B3	Surrounding jet tube	1
B4	Nozzle	1
B5	Central jet injection port	1
B6	Central jet measuring port	1
B7	Surrounding jet injection port	1
B8	Surrounding jet measuring port	1

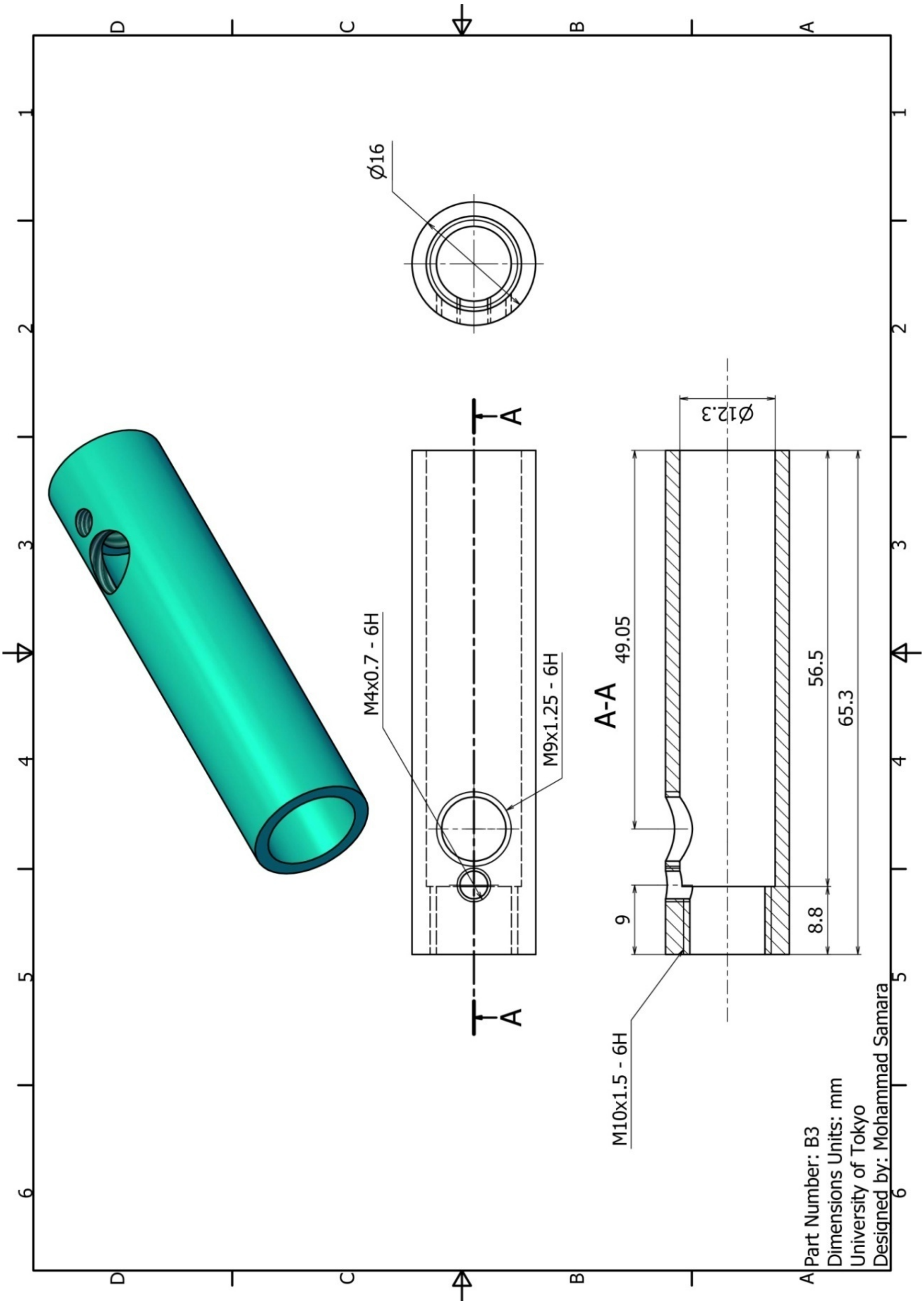
University of Tokyo
Designed by: Mohammad Samara

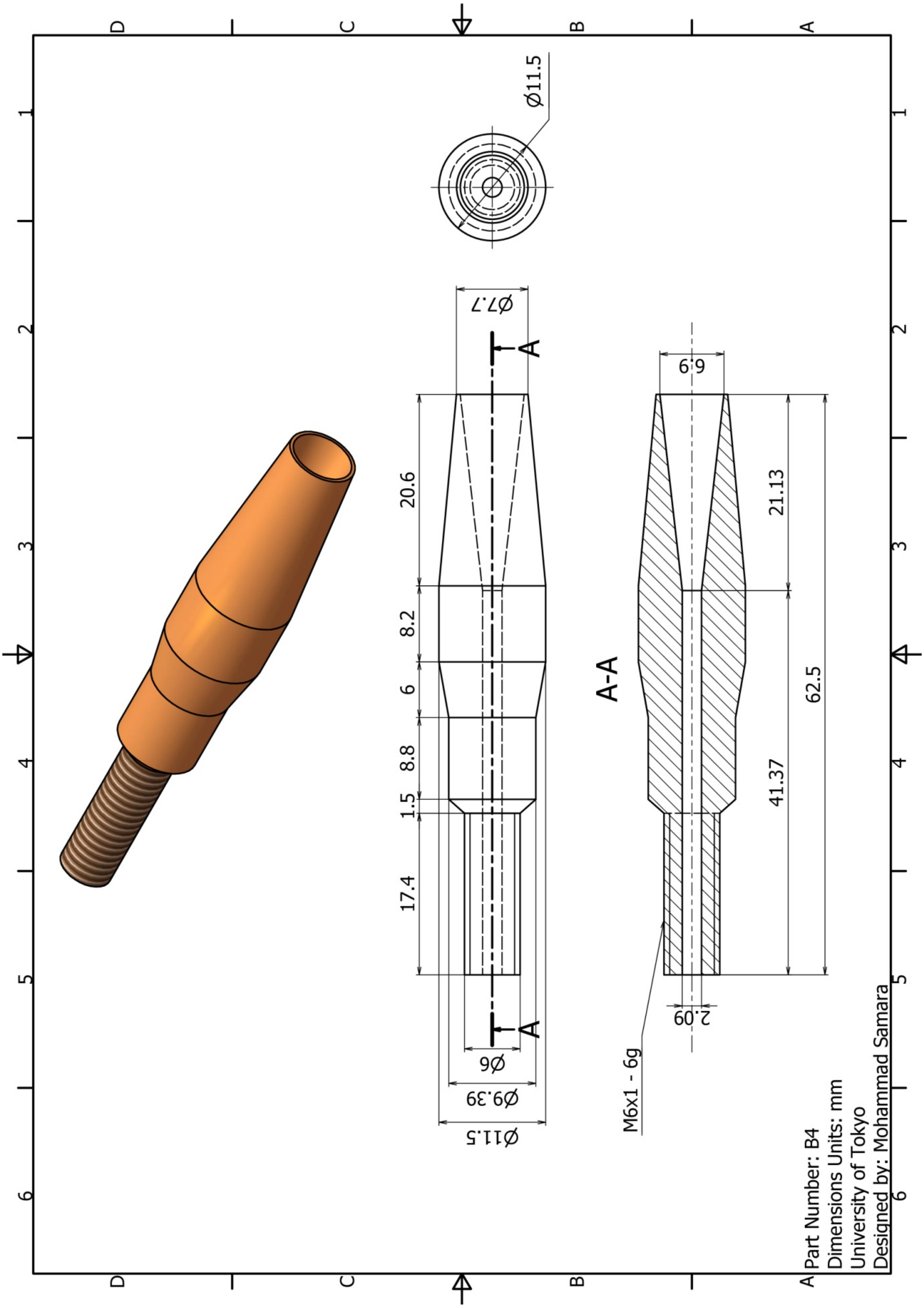


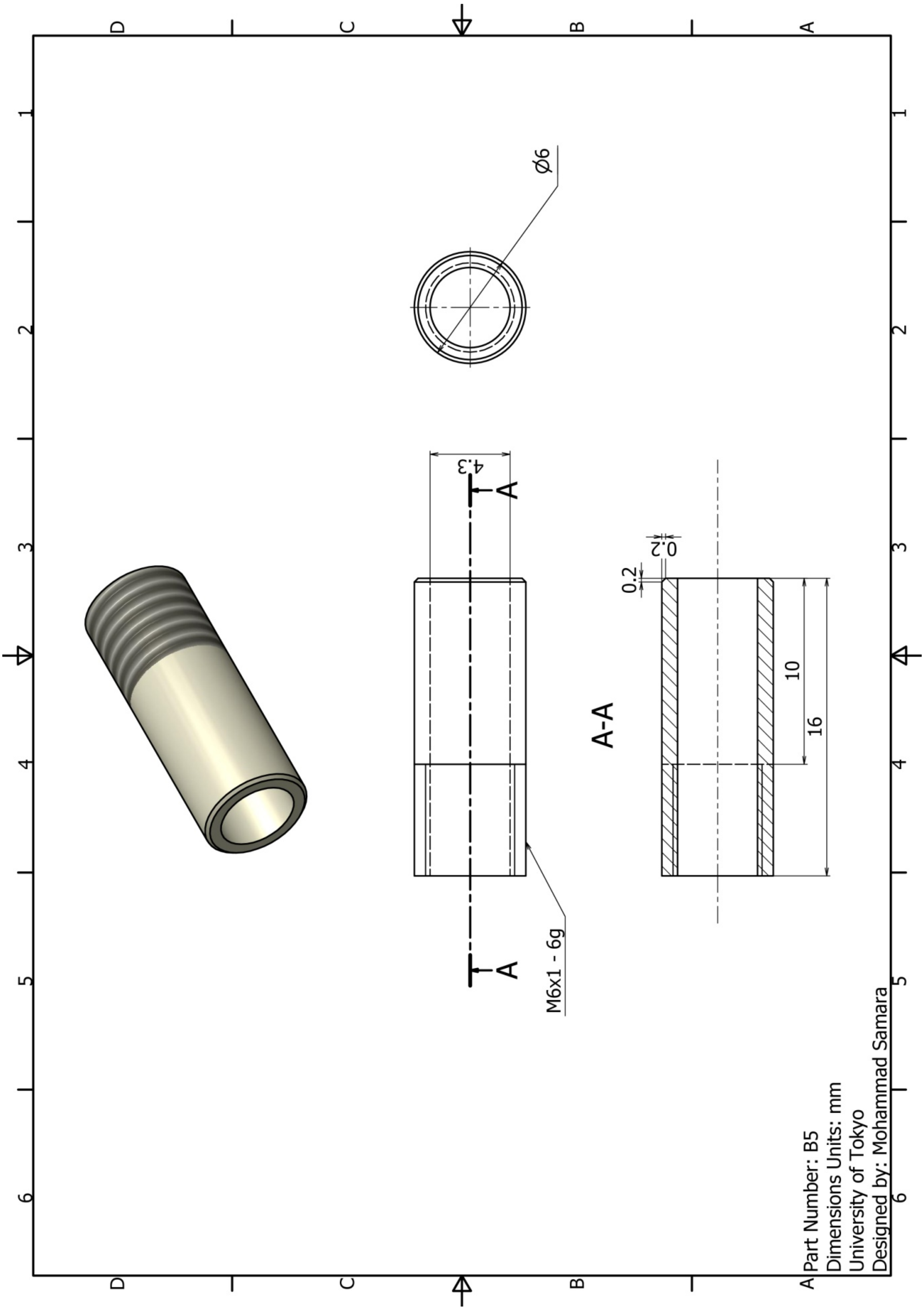
Ø16
Ø11
Ø4.8
B
B-B
M6x1 - 6H
12.9
2.7
8.5
3.2
37.3
M11x1.5 - 6g
6.3
Ø10.6
Ø16
Part Number: B1
Dimensions Units: mm
University of Tokyo
Designed by: Mohammad Samara



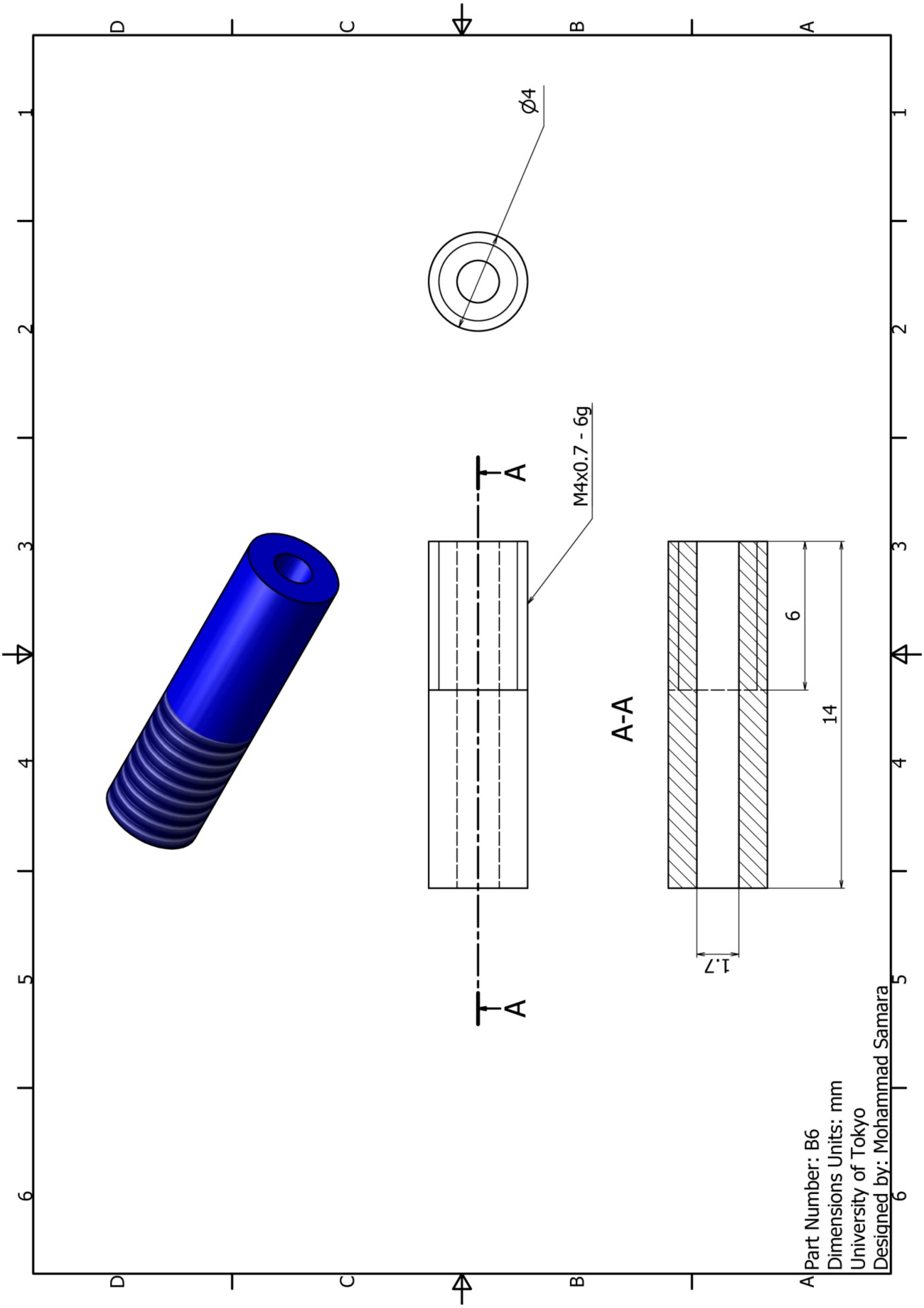
Ø16
M6x1 - 6H
M4x0.7 - 6H
M11x1.5 - 6H
Ø10.6
14.55
28.4
A-A
6.85
1.67
24.13
35.2
9.4
6.8
M6x1 - 6H
M10x1.5 - 6g
Ø6
Ø9.7
A
A
Part Number: B2
Dimensions Units: mm
University of Tokyo
Designed by: Mohammad Samara

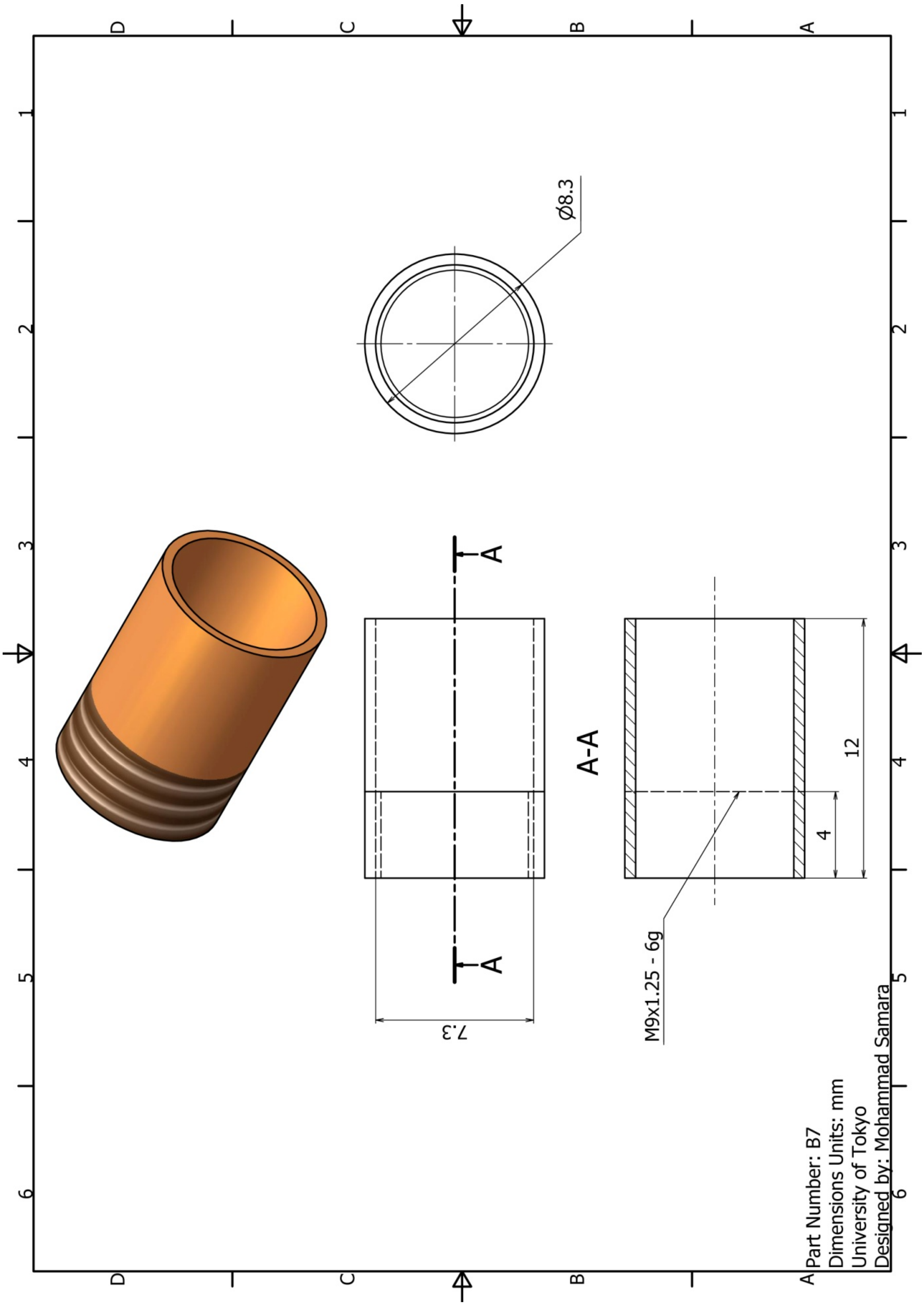

Ø16
Ø12.3
A
A
A-A
M4x0.7 - 6H
M9x1.25 - 6H
49.05
56.5
65.3
9
8.8
M10x1.5 - 6H
Part Number: B3
Dimensions Units: mm
University of Tokyo
Designed by: Mohammad Samara

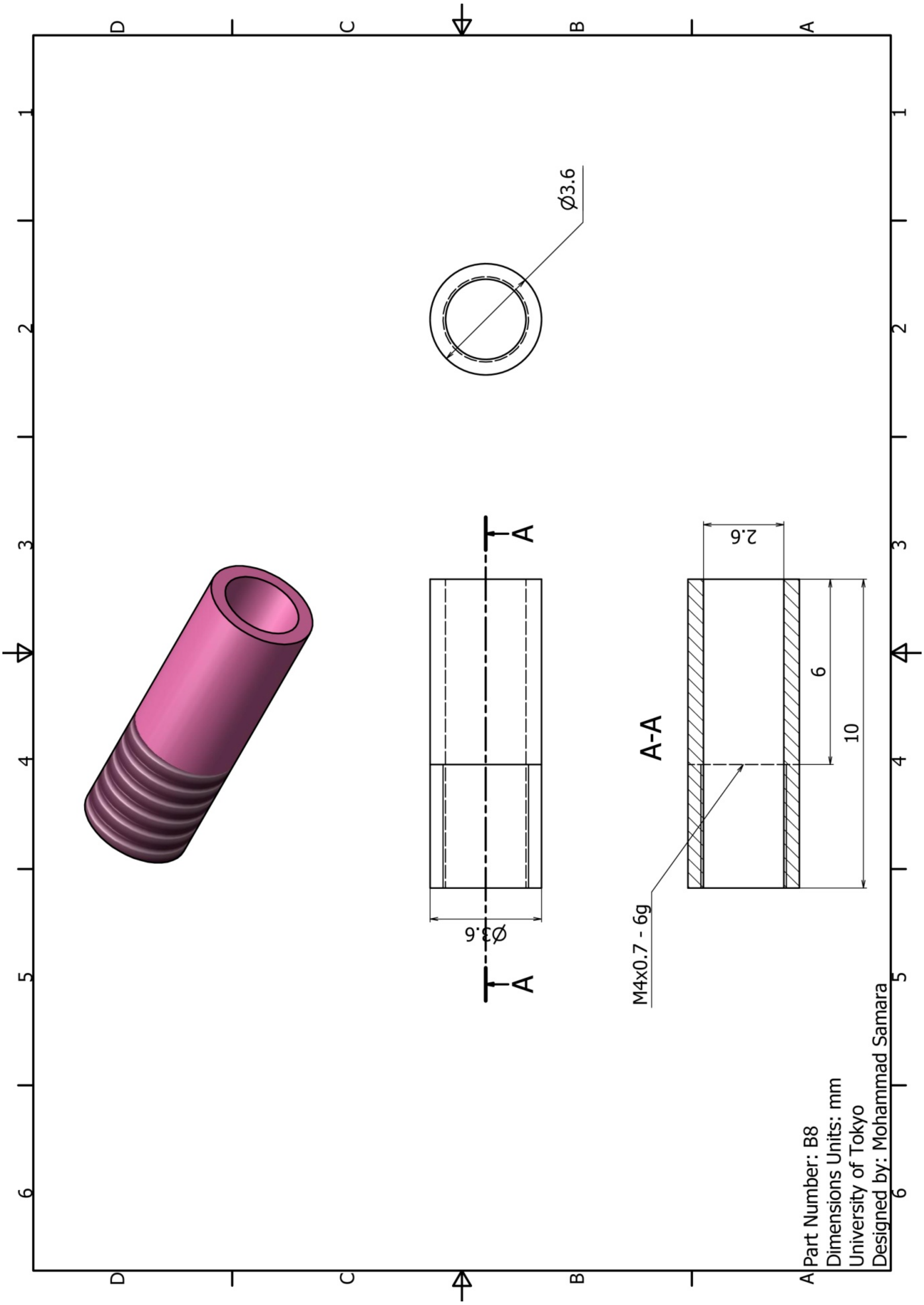

A-A
Ø11.5
Ø7.7
20.6
8.2
6
8.8
1.5
17.4
Ø6
Ø9.39
Ø11.5
6.9
21.13
62.5
41.37
2.09
M6x1 - 6g
Part Number: B4
Dimensions Units: mm
University of Tokyo
Designed by: Mohammad Samara



Ø6
4.3
A
A
A-A
0.2
0.2
10
16
M6x1 - 6g
Part Number: B5
Dimensions Units: mm
University of Tokyo
Designed by: Mohammad Samara


Ø4
M4x0.7 - 6g
A
A
A-A
6
14
1.7
Part Number: B6
Dimensions Units: mm
University of Tokyo
Designed by: Mohammad Samara


Ø8.3
A
A
A-A
7.3
12
4
M9x1.25 - 6g
Part Number: B7
Dimensions Units: mm
University of Tokyo
Designed by: Mohammad Samara


Ø3.6
A
A
Ø3.6
A-A
2.6
6
10
M4x0.7 - 6g
Part Number: B8
Dimensions Units: mm
University of Tokyo
Designed by: Mohammad Samara

Appendix II

Chamber Temperature Effect

The purpose of studying the current single jet flow in a hypersonic free stream; is to understand the fundamental characteristics of jet and free stream interaction, by studying an underexpanded supersonic axisymmetric jet from the tail of an experimental body with a simple axisymmetric shape; along with studying the chamber temperature.

Governing equations

Compressible axisymmetric Navier Stokes equations has been solved, the equations has an inviscid part, viscous part and a source term in order to make it axisymmetric, the solver is a generalized 2-D axisymmetric equations solver that solves the inviscid part of the Compressible axisymmetric Navier Stokesequations by using Liou'sall speed AUSM+up scheme, with a 3rd-ordered upwind biased MUSCL interpolation along with an Entropy fixation option. The 2nd ordered central difference method have been used for solving the viscous terms.The solver also utilizes the 3 step TVD RungeKutta for time marching.

Boundary conditions

Boundary conditions that is imposed in the numerical simulation are wall, inlet, outlet, axisymmetric, and a free stream conditions, such boundaries can be seen in the figure below which represent the experimental model chamber and Convergent-divergent nozzle along with the outer jet flow area.

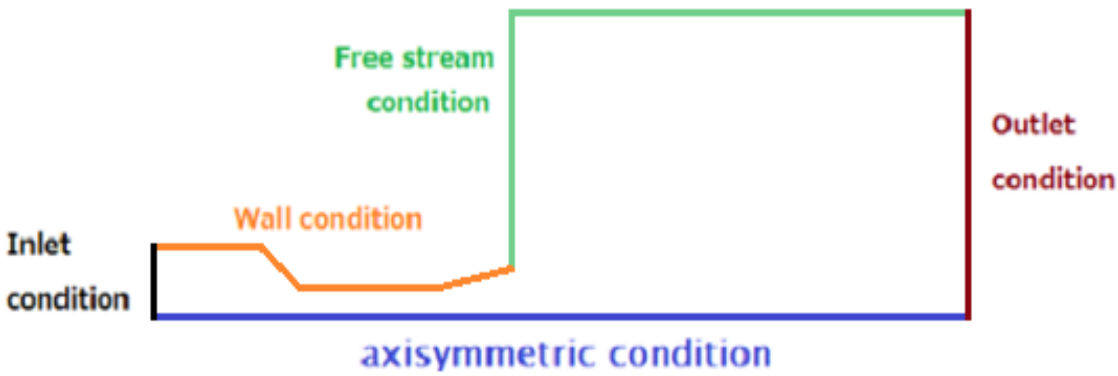


Figure A.2.1 Numericalmodel boundary conditions

Computational domain

The computational domain uses a structured two dimensional mesh, multigrid, multizone domain, the coupled grids can be seen in Figure 4.5, and the zones can be seen in Figure 4.6, for Zone A it has (mx = 301) x (my = 51) , and Zone B:(mx = 101) x (my = 201), the overall computational domain has a minimum mesh size of the order of (10-6), a mesh refinement near the walls, as well as when there is a change in the walls direction as it seen in Figure A.1.2.

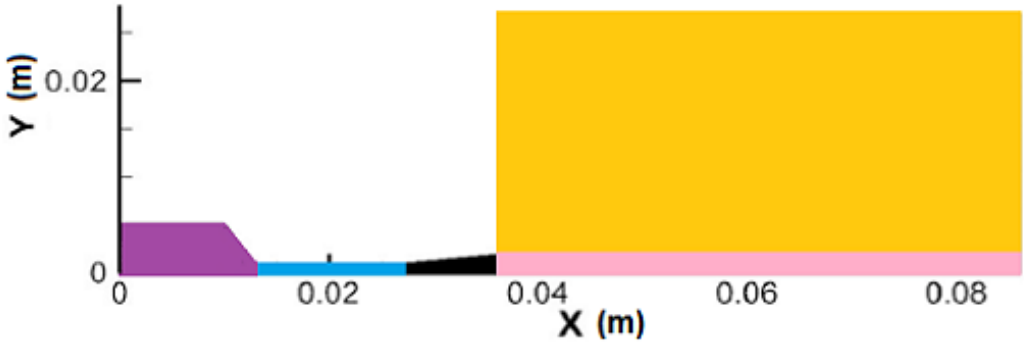


Figure A.2.2 Computational domains

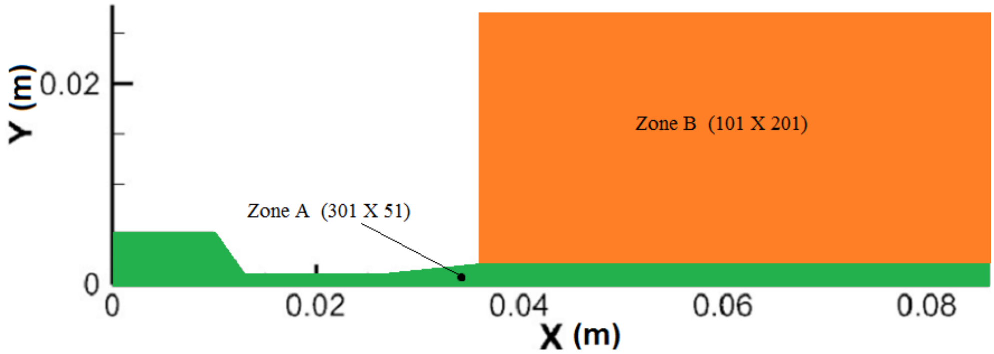


Figure A.2.3 Zoning of computational domain

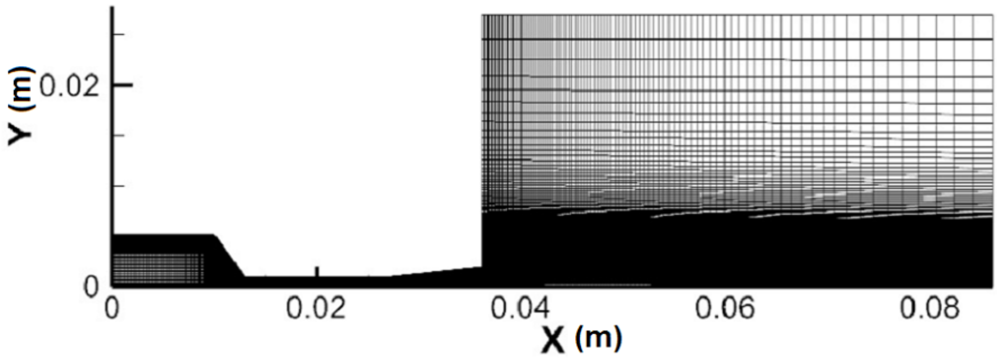


Figure A.2.4 Generated Mesh

This section is dedicated to study the effect of using hot jet flow instead cold flow for simulating an engine exhaust jets (so far been discussed and demonstrated by velocity gradient in figure A..2.5), the importance of this study lies in the easiness of the use of cold flow in the experiment, meanwhile the use of a hot air flow to simulate the exhaust can be difficult and much more dangers and expensive, taking what has been mentioned in consideration this study deals with the effect of jet flow temperature on the computational values of velocity, density, Mach number, and pressure, along with the exhaust jet shape.

The computation uses a Mach 7 free stream, with a static pressure of 950 kPa, and a static temperature of 55.0 K, the model nozzle walls is set to be 300 K, for the jet chamber pressure it is set to be 688,412 kPa with a temperatures of 300, 1000, and 3000 K. The model uses a multiple grid system with a multi computational zones presented before as the single jet flow without taking the effect of the hypersonic body into consideration.

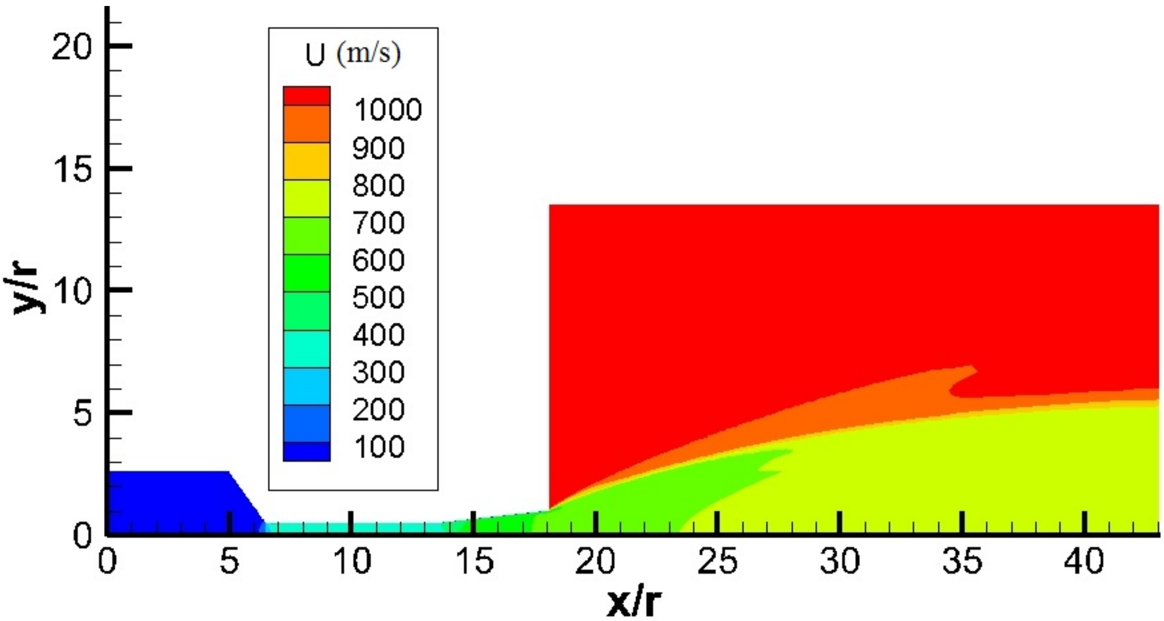


Figure A.2.5 300 K, jet stagnation temperature, effect on jet flow axial velocity contour.

The effect of changing the chamber total temperatures, is examined by varying the chamber temperature of the values of 300, 1000, and 3000 K, the figures below (figures A.2.6, and A.2.7) shows an expected effect of changing the chamber temperature on the velocity of the jet flow, that can be noted that as we increase the jet temperature the jet flow velocity will dramatically increase. One thing to note that in the case of chamber condition of temperature of 3000 K, and a pressure of 688,412 kPa, can represent a scramjet engine exhaust outlet condition.

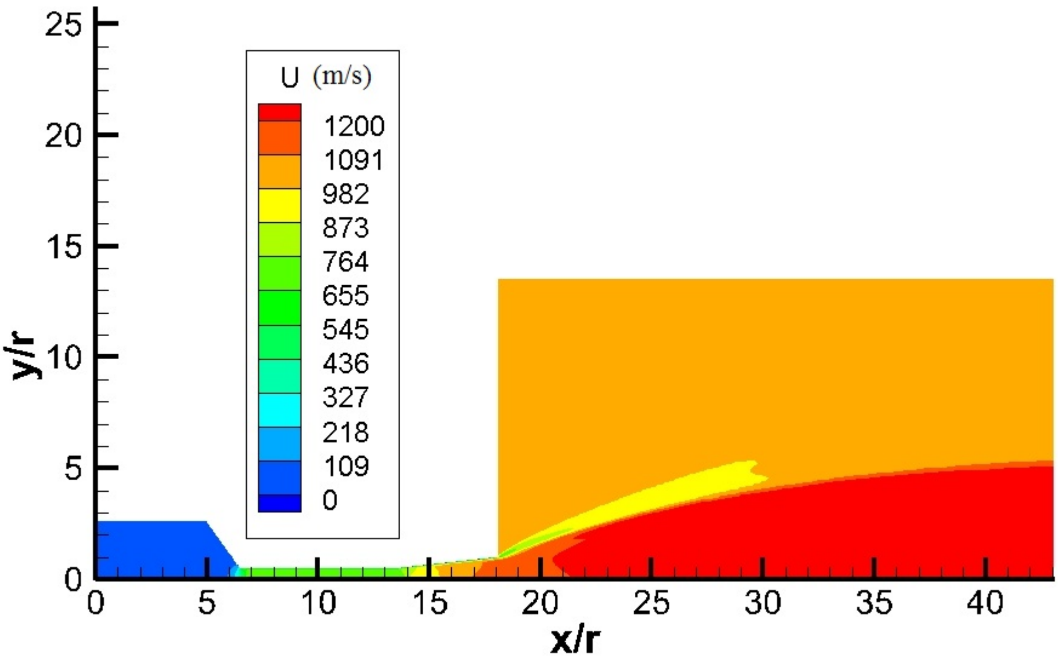


Figure A.2.6 1000 K, jet stagnation temperature, effect on jet flow axial velocity contour

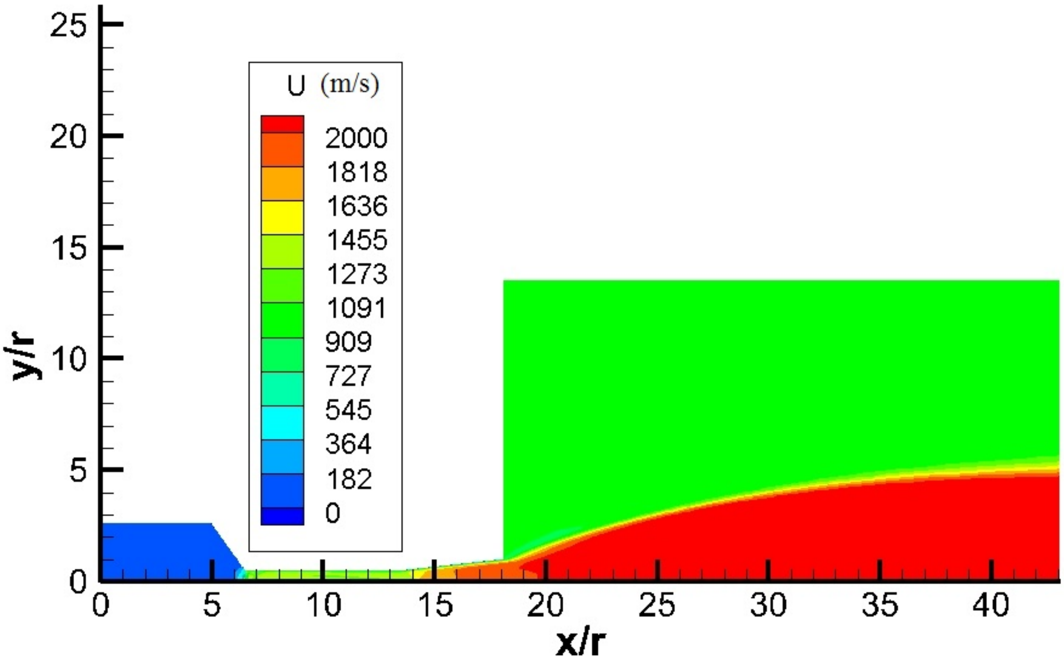


Figure A.2.7 3000 K, jet stagnation temperature, effect on jet flow axial velocity contour

The effect of jet temperature on the jet flow density is represented by figures A.2.8 , A.2.9, and A.2.10, in which can be noted that the use of cold jet flow can result in a more steep density gradient at the jet flow boundary, such gradient of density is important in detecting the jet boundary with the use of schlieren imaging system, that the acquired schlieren images will show the gradient of density of the flow, in an other words acquiring a steeper density gradients at the jet boundary will result of a more specific identification of the jet flow boundaries by using the schlieren imaging system.

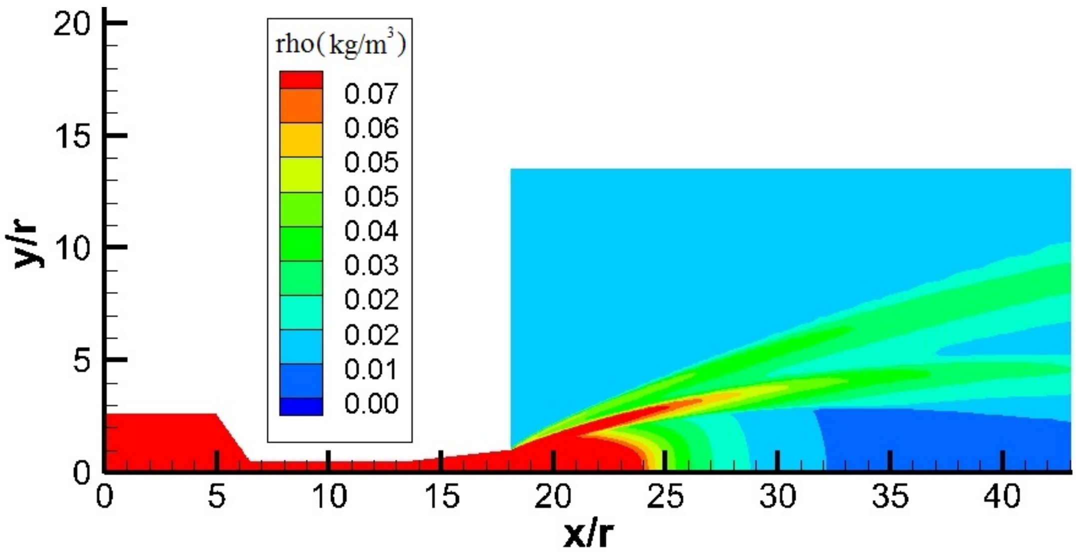


Figure A.2.8 300 K, jet stagnation temperature, effect on jet flow density contour

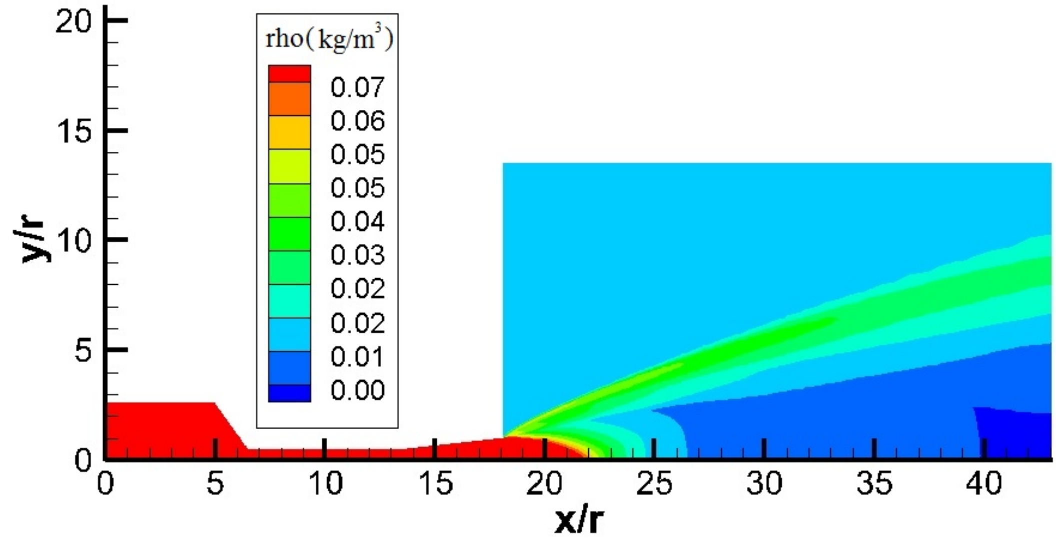


Figure A.2.9 1000 K, jet stagnation temperature, effect on jet flow density contour

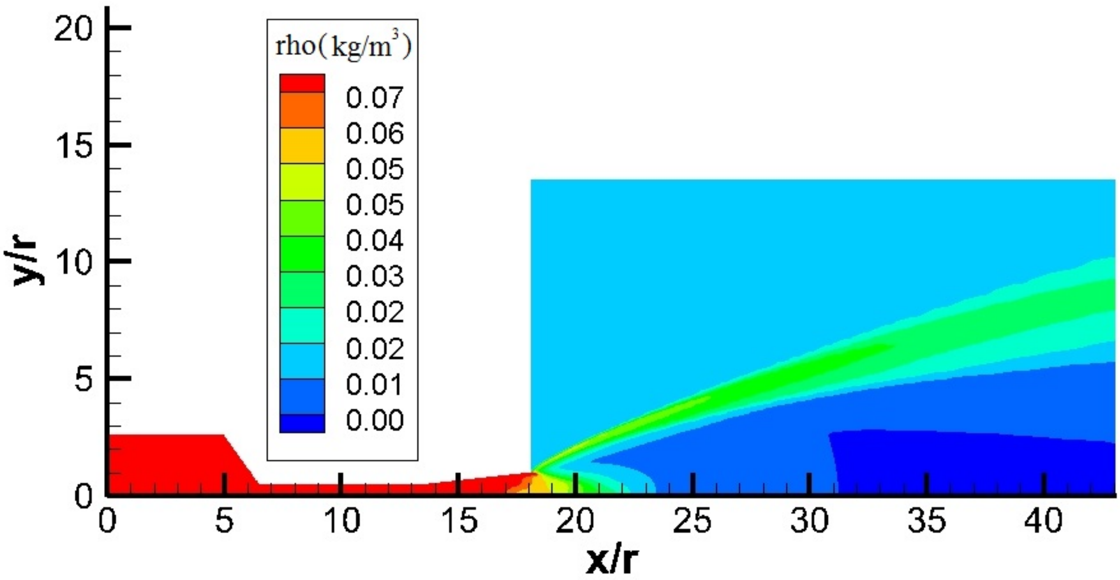


Figure A.2.10 3000 K, jet stagnation temperature, effect on jet flow density

The pressure contour is the most important to have a high similarity between the hot exhaust flow, and the cold flow; such importance came as a result of the role of pressure in creating lift, moments, and thrust in the case of the use of a ramp in which such forces will come as a result of the applied pressure on the ramp surface. The current study shows a very high similarity in the contours of pressure at the flows field of different temperatures, such result can be seen by examining figures A.2.11, A.2.12, and A.2.13.

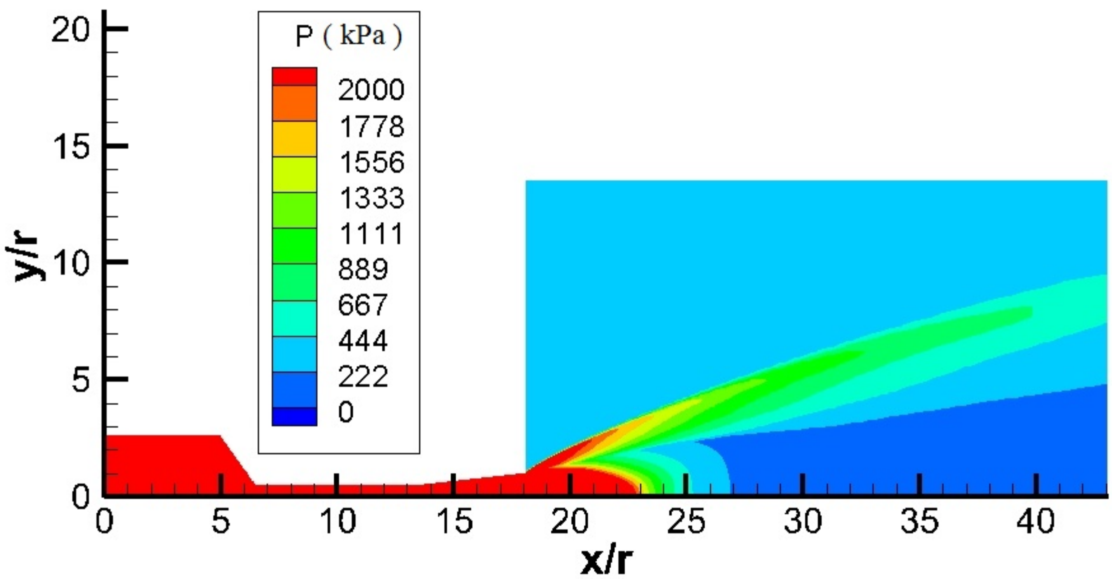


Figure A.2.11 300 K, jet stagnation temperature, effect on jet flow pressure contour

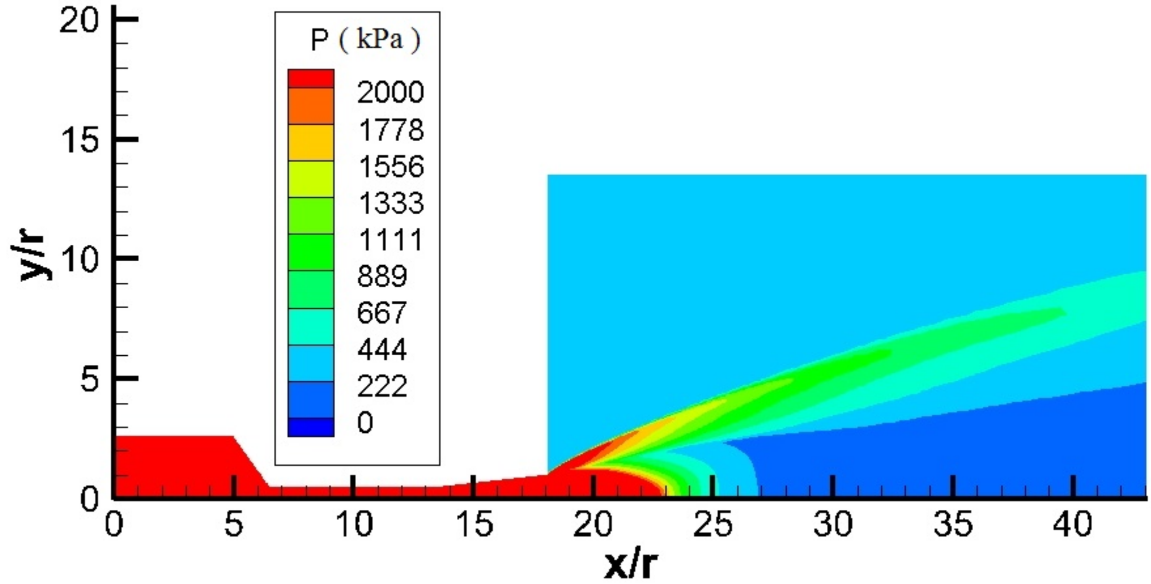


Figure A.2.12 1000 K, jet stagnation temperature, effect on jet flow pressure contour

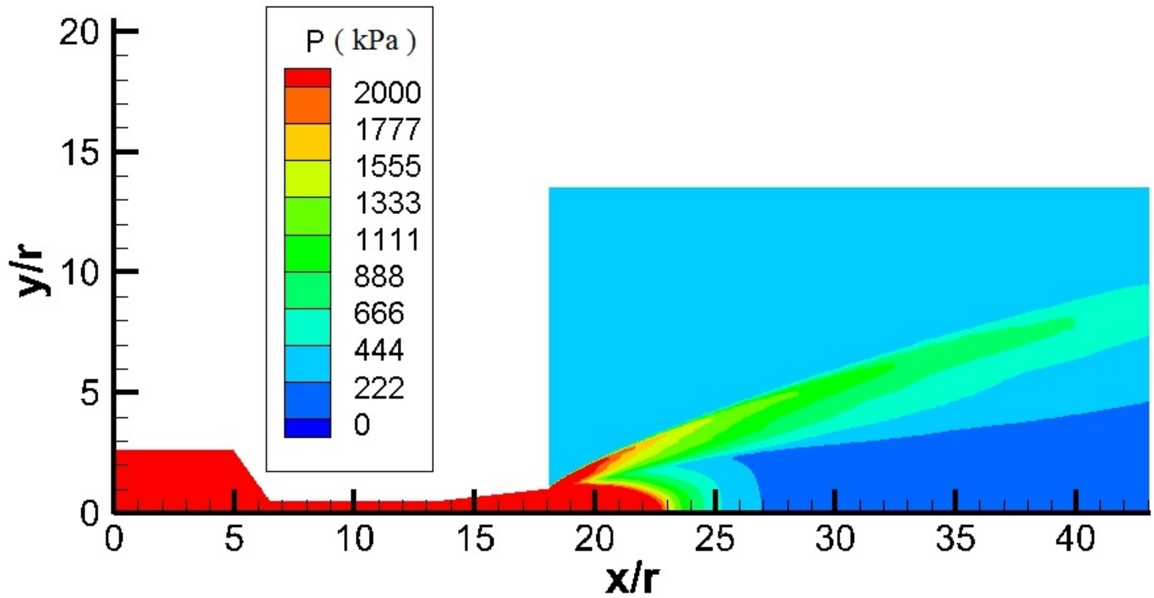


Figure A.2.13 3000 K, jet stagnation temperature, effect on jet flow pressure contour

It can be noted form examining the CFD results, is that the Mach number contour plot and in the cases of changing the jet stagnation temperature have no effect on the Mach number contour, such result can be seen in figures A.2.14, A.2.15, and A.2.16.

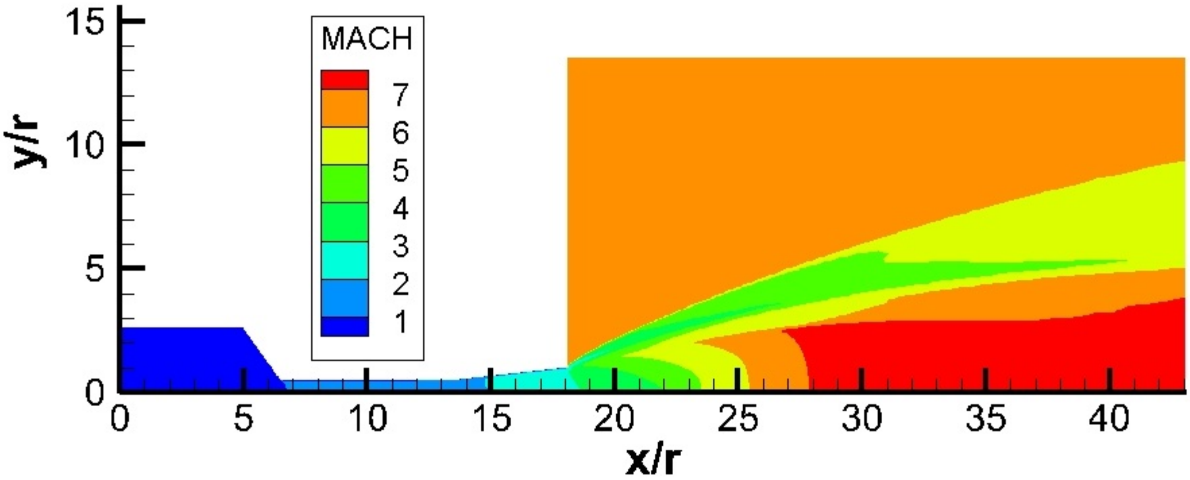


Figure A.2.14 300 K, jet stagnation temperature, effect on jet flow Mach number contour

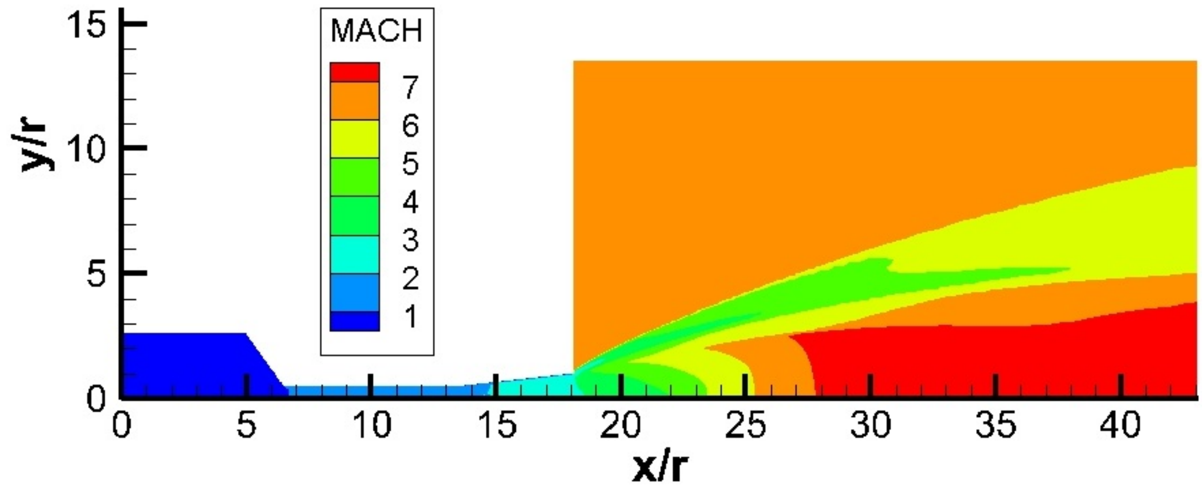


Figure A.2.15 1000 K, jet stagnation temperature, effect on jet flow Mach number contour

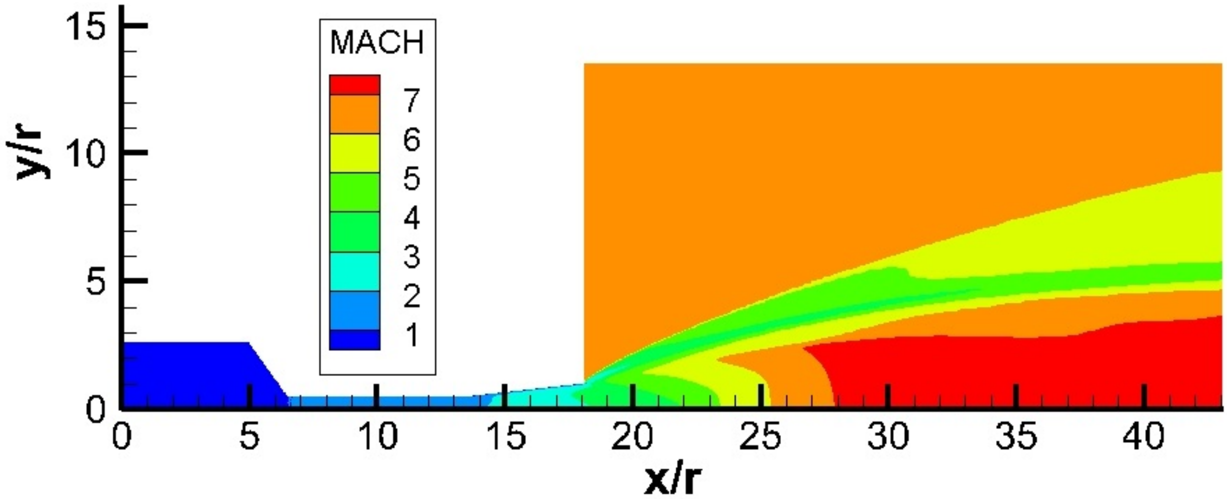


Figure A.2.16 3000 K, jet stagnation temperature, effect on jet flow Mach number contour

Doctoral Course Publications

Conference Papers:

-Experimental and Numerical Investigation on Coaxial Jets Exhausted from a Body in Hypersonic Flow, 13th International Symposium on Fluid Control, FLUCOME2015/ Measurement and Visualization, November 15-18, 2015, Doha, Qatar.

-Numerical and Experimental Investigation on Interaction between Flow around a Hypersonic Body and Supersonic Jet from its Tail, 31st International Symposium on Space Technology and Science (ISTS) June 3rd-9th, 2017, Matsuyama-city of Ehime-prefecture, Japan.

Journal Papers:

-Flow-Field and Performance Study of Coaxial Supersonic Nozzles Operating in Hypersonic Environment, International Review of Aerospace Engineering (I.RE.AS.E), Vol. xx, N. November 2019. **(Accepted)**